ANSYS Commands Reference

ANSYS Release 8.1

001972
April 2004

ANSYS Commands Reference

ANSYS Release 8.1

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Revision History

Number	Release	Date
001606	ANSYS 6.1	April 2002
001695*	ANSYS 7.0	October 2002
001788*	ANSYS 7.1	May 2003
001901*	ANSYS 8.0	October 2003
001972*	ANSYS 8.1	April 2004

* ANSYS Documentation on CD.

Trademark Information

Table of Contents

List of Tables

Chapter 1: About This Manual

Welcome to the *ANSYS Commands Reference*. This manual contains a complete dictionary of detailed ANSYS command descriptions, arranged in alphabetical order for ease of access. It is the definitive reference for correct command usage, providing comprehensive specifications for every argument of every ANSYS command.

1.1. Guide to the ANSYS Documentation

This manual is just one part of the ANSYS product documentation set, which includes the *ANSYS Operations Guide*, the *ANSYS Modeling and Meshing Guide*, the *ANSYS Basic Analysis Guide*, the *ANSYS Advanced Analysis Techniques Guide*, specific analysis guides, the *ANSYS Commands Reference*, the *ANSYS Elements Reference*, and the *ANSYS, Inc. Theory Reference*. Taken together, these manuals provide descriptions of the procedures, commands, elements, and theoretical details needed to use the ANSYS program. A brief description of the information found in each of the manuals follows:

ANSYS Operations Guide: Describes basic ANSYS operations (starting, stopping,etc.) and use of the graphical user interface (GUI).

ANSYS Modeling and Meshing Guide: Explains how to build a finite element model and mesh it.

ANSYS Basic Analysis Guide: Describes tasks that apply to any type of analysis, including applying loads to a model, obtaining a solution, and using the ANSYS program's graphics capabilities to review results.

ANSYS Advanced Analysis Techniques Guide: Discusses techniques commonly used for complex analyses or by experienced ANSYS users, including design optimization, submodeling, and substructuring.

ANSYS Structural Analysis Guide: Describes how to perform structural analyses: static, modal, harmonic, transient, spectrum, buckling, nonlinear, contact, fracture, composite, fatigue, p-method, explicit dynamics, and beam.

ANSYS Thermal Analysis Guide: Describes how to do steady-state or transient thermal analyses.

ANSYS Low-Frequency Electromagnetic Analysis Guide: Explains techniques for doing transient, static, or harmonic magnetic analyses; current conduction analysis; and electrostatic and circuit analysis.

ANSYS Fluids Analysis Guide: Describes how to perform laminar or turbulent, thermal or adiabatic and compressible or incompressible flow analysis and how to do multiple species transport and volume of fluid analyses.

ANSYS Coupled-Field Analysis Guide: Explains how to do indirect and direct coupled-field analysis (an analysis that accounts for the interaction between two or more engineering disciplines).

ANSYS Commands Reference: Describes the syntax of all ANSYS commands and is organized alphabetically.

ANSYS Elements Reference: Describes the ANSYS element library: figures, input data, output data, and other details of all ANSYS elements.

ANSYS, Inc. Theory Reference: Describes theory and ANSYS implementation of all ANSYS features: finite elements, solvers, postprocessors, elements, etc. This manual describes the relationship between input data and output results produced by the program and is essential for a thorough understanding of how the program functions.

All of the above manuals are available online through the ANSYS Help System, which can be accessed either as a stand-alone system or from within the ANSYS program.

NOTICE

The following manuals provide the complete specification of the ANSYS program referenced in your license agreement:

- *ANSYS Basic Analysis Guide*
- *ANSYS Advanced Analysis Techniques Guide*
- *ANSYS Operations Guide*
- *ANSYS Modeling and Meshing Guide*
- *ANSYS Structural Analysis Guide*
- *ANSYS Thermal Analysis Guide*
- *ANSYS Low-Frequency Electromagnetic Analysis Guide*
- *ANSYS Fluids Analysis Guide*
- *ANSYS Coupled-Field Analysis Guide*
- *ANSYS Elements Reference*
- *ANSYS Commands Reference*
- *ANSYS, Inc. Theory Reference*

This specification describes how to use the program, input required for commands and elements, and how the input data relate to the output.

Other documents (such as the *ANSYS Tutorials* and seminars, various *Supplements*, etc.) may be referenced by or listed in the ANSYS manuals listed above. However, these other documents are offered solely as aids, and do not undergo the same rigorous verification that the ANSYS manuals listed above do. *No documents other than the manuals listed above are considered to be part of the formal program specification as stated in your license agreement.*

1.2. The ANSYS Commands Reference

The *ANSYS Commands Reference* is intended to give you information on individual ANSYS commands. Although this manual also contains lists of commands arranged by functional grouping, this manual is not intended to be your primary source of procedural information -- look in the appropriate analysis guides for introductory and procedural guidelines concerning when and where to use commands.

Command references given in the various analysis guides refer to the command descriptions given in Chapter 3, "Command Dictionary" of this manual. These descriptions are organized in dictionary form to allow you to look up the commands conveniently. The alphabetical ordering of commands that begin with a star (*****) or a slash (**/**) ignores those symbols for ordering purposes.

Once you have reviewed a command in the *ANSYS Commands Reference*, you should also read the theoretical description of the command as given in the *ANSYS, Inc. Theory Reference*, if indicated.

1.2.1. Conventions Used in this Manual

ANSYS manuals use the following conventions to help you identify various types of information:

Type style or text	Indicates
BOLD	Uppercase, bold text indicates command names (such as **K**,**DDELE**) or elements (LINK1).

Type style or text	Indicates
Bold>Bold	Bold text in mixed case indicates a GUI menu path, which is a series of menu picks used to access a command from the GUI. One or more angle brackets (>) separate menu items in a menu path. Frequently in text, an ANSYS command is followed by its GUI equivalent in parentheses: the ***GET** command (**Utility Menu> Parameters> Get Scalar Data**)
ITALICS	Uppercase italic letters indicate command arguments for numeric values (such as *VALUE*, *INC*, *TIME*). On some commands, non-numeric convenience labels (for example, ALL and P) can also be entered for these arguments.
Italics	Mixed case italic letters indicate command arguments for alphanumeric values (for example, *Lab* or *Fname*). The manual also uses italic text for emphasis.
TYPEWRITER	Typewriter font indicates command input listings and ANSYS output listings.
Note--	This text introduces note paragraphs. A note contains information that supplements the main topic being discussed.

Note — Any mention of a command or element name in this volume implies a reference to the appropriate command or element description (in the *ANSYS Commands Reference* or *ANSYS Elements Reference* manuals, respectively) for more detailed information.

Often it is necessary to refer to another command within a text description. Interpret the string [**CMD**] as "see the **CMD** command." For example, " ... the element coordinate system [**ESYS**] should be used to ...," means you can refer to the **ESYS** command for information related to the element coordinate system. Where several commands are referenced, the form [**CMD1**, **CMD2**, etc.] is used. In some cases only the primary command of a related group may be referenced.

1.2.1.1. Product Codes

Below and to the right of the short description of each command, you will see a list of product codes. These codes represent the products in the ANSYS Family of Products. The command is valid only for those products whose symbols are listed. A command that is valid in the entire set of products would have the following list of products:

MP ME ST DY <> PR EM <> FL PP ED

The codes represent each of the products in the ANSYS suite of products:

Code	Product	Code	Product
MP	ANSYS Multiphysics	EM	ANSYS Emag - Low Frequency
ME	ANSYS Mechanical	EH	ANSYS Emag - High Frequency
ST	ANSYS Structural	FL	ANSYS FLOTRAN
DY	ANSYS LS-DYNA	PP	ANSYS PrepPost
VT	ANSYS DesignXplorer VT	ED	ANSYS ED
PR	ANSYS Professional	DP	ANSYS LS-DYNA PrepPost

Note — While DP (ANSYS/LS-DYNA PrepPost) does not appear as a unique product code in the product listings for commands and elements, it does appear as a separate product in other places in the manuals.

For a list of the features included in each product, see Section 1.2.3: ANSYS Product Capabilities.

If the symbol for a product does not appear, then that command is either not valid or not applicable in the corresponding product, and should not be issued when using that product. For example, if the PR and FL symbols are not listed, the pertinent command is not valid in the ANSYS Professional or ANSYS FLOTRAN products, but is valid in each of the remaining ANSYS products.

1.2.2. Applicable ANSYS Products

This manual applies to the following ANSYS products:

ANSYS Multiphysics (includes all structural, thermal, electromagnetics, and computational fluid dynamics (CFD) capabilities, excludes explicit dynamics)
ANSYS Mechanical (includes all structural and thermal capabilities; excludes electromagnetics, CFD, and explicit dynamics capabilities)
ANSYS Structural (includes all structural linear and nonlinear capabilities)
ANSYS Professional
ANSYS Emag (Low Frequency and High Frequency)
ANSYS FLOTRAN
ANSYS LS-DYNA
ANSYS LS-DYNA PrepPost
ANSYS PrepPost
ANSYS ED

Some command arguments and element KEYOPT settings have defaults in the derived products that are different from those in ANSYS Multiphysics. These cases are clearly documented under the "Product Restrictions" section of the affected commands and elements. If you plan to use your derived product input file in ANSYS Multiphysics, you should explicitly input these settings in the derived product, rather than letting them default; otherwise, behavior in ANSYS Multiphysics will be different.

Note — While ANSYS Connection, Parallel Performance for ANSYS, and ANSYS LSF/Batch are included as part of the ANSYS release distribution, they are separately-licensed products. Consult your ASD if you want to install and run any of the separately-licensed products at your site.

1.2.3. ANSYS Product Capabilities

Here is a complete list of engineering capabilities and the various ANSYS products in which these capabilities can be found.

Capability	Product Availability
Structural Linear	
Static	MP ME ST <> <> PR <> <> <> <> ED
Transient	MP ME ST DY <> PR <> <> <> <> ED
Substructuring	MP ME ST <> <> <> <> <> <> <> ED
Structural Nonlinear	
Static	MP ME ST <> <> PR <> <> <> <> ED
Transient	MP ME ST DY <> <> <> <> <> <> ED
Geometric	MP ME ST DY <> PR <> <> <> <> ED

Capability	Product Availability
Material	MP ME ST DY <> <> <> <> <> ED
Element	MP ME ST DY <> PR <> <> <> <> ED
Structural Contact/Common Boundaries	
Surface to Surface	MP ME ST DY <> PR <> <> <> <> ED
Node to Surface	MP ME ST DY <> <> <> <> <> <> ED
Node to Node	MP ME ST <> <> PR <> <> <> <> ED
Pretension (bolts, etc.)	MP ME ST <> <> <> <> <> <> <> ED
Interface (gaskets)	MP ME ST <> <> <> <> <> <> <> ED
Structural Dynamic	
Modal	MP ME ST <> <> PR <> <> <> <> ED
Spectrum	MP ME ST <> <> PR <> <> <> <> ED
Harmonic	MP ME ST <> <> PR <> <> <> <> ED
Random Vibration	MP ME ST <> <> <> <> <> <> <> ED
Structural Buckling	
Linear	MP ME ST <> <> PR <> <> <> <> ED
Nonlinear	MP ME ST DY <> <> <> <> <> <> ED
Thermal Analysis	
Steady State	MP ME <> <> <> PR <> <> FL <> ED
Transient	MP ME <> <> <> PR <> <> FL <> ED
Conduction	MP ME <> <> <> PR <> <> FL <> ED
Convection	MP ME <> <> <> PR <> <> FL <> ED
Radiation	MP ME <> <> <> PR <> <> FL <> ED
Phase Change	MP ME <> <> <> PR <> <> <> <> ED
CFD Analysis	
Steady State	MP <> <> <> <> <> <> <> FL <> ED
Transient	MP <> <> <> <> <> <> <> FL <> ED
Incompressible	MP <> <> <> <> <> <> <> FL <> ED
Compressible	MP <> <> <> <> <> <> <> FL <> ED
Laminar	MP <> <> <> <> <> <> <> FL <> ED
Turbulent	MP <> <> <> <> <> <> <> FL <> ED
Forced Convection	MP <> <> <> <> <> <> <> FL <> ED
Natural Convection	MP <> <> <> <> <> <> <> FL <> ED

Capability	Product Availability
Conjugate Heat Transfer	MP <> <> <> <> <> <> <> FL <> ED
Radiation Heat Transfer	MP <> <> <> <> <> <> <> FL <> ED
Multiple Species Transport	MP <> <> <> <> <> <> <> FL <> ED
Newtonian Viscosity Model	MP <> <> <> <> <> <> <> FL <> ED
Non-Newtonian Viscosity Model	MP <> <> <> <> <> <> <> FL <> ED
Rotating Frames of Reference	MP <> <> <> <> <> <> <> FL <> ED
Distributed Resistances & Sources	MP <> <> <> <> <> <> <> FL <> ED
2-D Free Surface by VOF Method	MP <> <> <> <> <> <> <> FL <> ED
Deformable Meshes (ALE Formulation)	MP <> <> <> <> <> <> <> FL <> ED
Electromagnetic - Low Frequency	
Electrostatics	MP <> <> <> <> <> EM <> <> <> ED
Magnetostatics	MP <> <> <> <> <> EM <> <> <> ED
LF-Electromagnetics	MP <> <> <> <> <> EM <> <> <> ED
Current Conduction	MP <> <> <> <> <> EM <> <> <> ED
Circuit Analysis &Coupling	MP <> <> <> <> <> EM <> <> <> ED
Harmonic	MP <> <> <> <> <> EM <> <> <> ED
Transient	MP <> <> <> <> <> EM <> <> <> ED
Electromagnetics - High Frequency	
Modal 2D-Waveguide/Cavity	MP <> <> <> <> <> <> EH <> <> ED
Modal 3D-Waveguide/Cavity	MP <> <> <> <> <> <> EH <> <> ED
Scattering	MP <> <> <> <> <> <> EH <> <> ED
Harmonic	MP <> <> <> <> <> <> EH <> <> ED
Perfect Electric & Magnetic Conductors	MP <> <> <> <> <> <> EH <> <> ED
Impedance Boundaries	MP <> <> <> <> <> <> EH <> <> ED
Perfectly Matched Absorber Boundaries	MP <> <> <> <> <> <> EH <> <> ED
Near & Far Electromagnetic Field Extension	MP <> <> <> <> <> <> EH <> <> ED
Antenna Radiation Patterns	MP <> <> <> <> <> <> EH <> <> ED
Radar Cross Section	MP <> <> <> <> <> <> EH <> <> ED
High Frequency Modal	MP <> <> <> <> <> <> EH <> <> ED
High Frequency AC Harmonic	MP <> <> <> <> <> <> EH <> <> ED
Field and Coupled-Field Analysis	

Capability	Product Availability
Acoustics	MP ME <> <> <> <> <> <> <> ED
Acoustics-Structural	MP ME <> <> <> <> <> <> <> ED
Electric-Magnetic	MP <> <> <> <> <> EM <> <> <> ED
Fluid-Structural	MP <> <> <> <> <> <> <> <> <> ED
Fluid-Thermal	MP <> <> <> <> <> <> <> FL <> ED
Electromagnetic-Fluid	MP <> <> <> <> <> <> <> <> <> ED
Magnetic-Structural	MP <> <> <> <> <> <> <> <> <> ED
Electromagnetic-Thermal	MP <> <> <> <> <> <> <> <> <> ED
Piezoelectric	MP ME <> <> <> <> <> <> <> <> ED
Piezoresistive	MP <> <> <> <> <> <> <> <> <> ED
Thermal-Electric	MP ME <> <> <> PR <> <> <> <> ED
Thermal-Structural	MP ME <> <> <> PR <> <> <> <> ED
Electric-Electromagnetic-Thermal-Structural	MP ME <> <> <> <> <> <> <> <> ED
Solvers	
Iterative	MP ME ST <> <> PR EM EH FL <> ED
Sparse	MP ME ST <> <> PR EM EH <> PP ED
Frontal	MP ME ST <> <> PR EM EH <> PP ED
Explicit	<> <> <> DY <> <> <> <> <> <> ED
Preprocessing	
Solid Modeling	MP ME ST DY <> PR EM <> FL PP ED
Defeaturing	MP ME ST DY <> PR EM <> FL PP ED
IGES Geometry Transfer	MP ME ST DY <> PR EM <> FL PP ED
Meshing	MP ME ST DY <> PR EM <> FL PP ED
Loads and Boundary Conditions (Solid Model, Tabular, & Function)	MP ME ST DY <> PR EM EH FL PP ED
Postprocessing	
Contour Displays	MP ME ST DY <> PR EM EH FL PP ED
Vector Displays	MP ME ST DY <> PR EM EH FL PP ED
Isosurface Displays	MP ME ST DY <> PR EM EH FL PP ED
Slicing planes	MP ME ST DY <> PR EM EH FL PP ED
Particle Tracing	MP <> <> <> <> <> EM EH FL PP ED
Animation	MP ME ST DY <> PR EM EH FL PP ED

Capability	Product Availability
Results Listing	MP ME ST DY <> PR EM EH FL PP ED
Output (VRML, Postscript, TIFF)	MP ME ST DY <> PR EM EH FL PP ED
General Features	
HTML Report Generation	MP ME ST DY <> PR EM EH FL PP ED
Probabilistic Design System (PDS)	MP ME ST DY <> PR EM EH FL <> ED
Submodeling	MP ME ST <> <> PR EM EH FL PP ED
Optimization	MP ME ST DY <> PR EM EH FL <> ED
ANSDPS Parametric Design Language (APDL)	MP ME ST DY <> PR EM EH FL PP ED
Parametric Simulation	MP ME ST DY <> PR EM EH FL PP ED
Variational Technology	<> <> <> <> VT <> <> <> <> <> <>

1.2.4. Terminology

Various terms are used in the command descriptions throughout this manual. These terms are defined as follows:

Analysis - The set of input lines relating to a single problem. An analysis basically consists of three phases: the Preprocessing Phase, the Solution Phase, and the Postprocessing Phase.

Command - An instruction to supply data, or control, to the program. Commands usually begin with a prescribed name, followed by alphanumeric data. For example, the command **ET**, *ITYPE*, *Ename*, ... may be input as **ET**,1, BEAM3 stating that element type 1 is defined as the BEAM3 element. The uppercase argument name indicates that a numerical value is typically entered in that field, whereas an upper-lower case data label indicates that an alphanumeric value is typically entered in that field. Another command, for example, **/PREP7**, instructs the program to enter the PREP7 portion of the ANSYS program. All valid ANSYS commands are alphabetically listed in the Chapter 3, "Command Dictionary" of this manual. A command not recognized by the ANSYS program is defined as an "unknown command." Unknown commands are further processed in a macro search [***USE**] before being ignored. Commands may be indented on a line for clarity.

Data - Data may be numeric (real or integer), alphabetic, or alphanumeric (containing letters and numbers). Nonnumeric data should not contain special characters such as

```
!   @   #    $   %    ^   &   *   (    )   _    -   +
=   |   \   {   }   [   ]    "    '   /    <    >   ~    `
```

Numeric data may be input in a variety of ways as described in Section 1.3.1: Data Input. Some commands are switches of the form **Commandname**,*Key* where *Key* can be 0, NO or OFF to toggle the switch off; or 1, YES or ON to toggle the switch on.

Degree of Freedom - The degrees of freedom are the primary nodal unknowns determined by the analysis. They may be displacements, rotations, temperatures, pressures, voltages, etc. A degree of freedom is defined by a node number and a label, for example, 1 UX, 87 ROTZ, 4 TEMP, etc. Derived results, such as stresses, heat flows, etc. are computed from the degree of freedom results and are considered secondary unknowns.

Element types with unequal sets of degrees of freedom can be combined in a single structure. For example, a 2-D structure composed of 2-D solid elements (PLANE42) having two degrees of freedom (UX, UY) at each node and a 2-D beam element (BEAM3) having three degrees of freedom (UX, UY, ROTZ) at each node will have the

latter three degrees of freedom at the common nodes. Nodes which do not have a beam element attached will have only two degrees of freedom with ROTZ eliminated from the solution printout.

Field - The command name and data items entered on a command are separated into consecutive fields. A field is assumed to be as "wide" as the number of characters specified. A comma is used to end one field and begin the next.

Line - A line of input is a physical record read by the computer. Input lines are limited to 640 characters (including preceding blanks, commas, and any special characters). For file input, a line is represented by one 640 column data record. For interactive input, a line is the string of characters (640 maximum) entered before the RETURN key is pressed. Several commands may be put on one line as described in Section 1.3.1: Data Input. Blank lines are permitted for clarity.

Command name - Only the first four characters of any alphabetic (or alphanumeric) command name are interpreted by the program (except as noted for certain commands, such as **/POST*N***, **/AUX*N***. ***ENDDO**, etc.). The remaining characters of the field are ignored. Names shown with fewer than four characters are assumed to have blanks up through the fourth character as part of the name. For example, this sample **ET** command may be input as ,**ET**1,42 or **ET**,1,42 or **ET**,1,42, but not as **ETABCD**,1,42. Names may not contain special characters (as described above in Section 1.3.1: Data Input). If the command name is omitted, the name defaults to the name of the previous command, unless it was a slash (/) or star (*) command.

Postprocessing Phase - The set of ANSYS commands causing further processing of the solution output. These commands consist of commands from the POST1 and POST26 processors. The postprocessing phase may consist of input for several postprocessing sessions (in series).

Preprocessing Phase - The set of ANSYS commands related to defining the model. The preprocessing phase consists of input from the PREP7 processor.

Program - The collection of all processors (preprocessing, postprocessing, auxiliary, etc.) is called the ANSYS program.

Processor - A group of related functions, such as model definition (PREP7) or results examination (POST1).

Run - The collection of all ANSYS commands between the system level commands is called an ANSYS run (or session). A session may also consist of several analyses in series (separated by a **/CLEAR** command).

Solution Phase - The set of ANSYS commands which apply boundary conditions to the model created in preprocessing, then performs a solution for that set of boundary conditions. The solution phase may consist of several solutions in series, such as a static solution, followed by a modal solution, etc.

1.3. ANSYS Command Characteristics

1.3.1. Data Input

The data input for each command is described in this manual. Data should not be input in any undocumented field, nor should other than documented values be input in any field. Also, the data input described in this manual should not be used with any earlier version of the ANSYS program. Some features that allow easy input of data are free-format, nonrestrictive, and condensed input.

1.3.2. Free-Format Input

Free-format capability allows the user to input data in consecutive fields without having to space to each field. The comma (,) character effectively ends the field so that the next character will be input in the beginning of the

next field. A blank field is skipped by having no data item between the commas. Fields are assumed to be as wide as the number of characters specified. Input is converted to formatted fields when coded files are written (of width large enough (16 characters maximum) to minimize loss of accuracy).

Significant figures of output should not be expected to match that of input. Machine precision, rounding of numbers when writing internal scratch files, etc., tend to lower the precision during the analysis.

1.3.3. Nonrestrictive Data Input

Nonrestrictive data input allows the user to enter any form of data in a field and the ANSYS program will interpret it as required (integer, real, or alphabetic) or ignore it. Double precision is used throughout the program for all real numbers. Alphabetic data may be entered in upper or lower case. Lower case is internally converted to upper case within the program (except for case-sensitive applications, such as in comments, (text preceded by a !) titles, and file names.)

The following features are available with the nonrestrictive data input.

- No distinction is necessary between real and integer data.
- Data may be placed anywhere within the field.
- Real data input without a decimal point has the decimal point assumed after the right-most digit.
- Real number values input to integer data fields will be rounded to the nearest integer. The absolute value of integer data must fall between zero and 99,999,999.
- Exponents may be input in the field after the number. The E (or D) character must be used (upper or lower case). The sign after the E character is assumed to be the sign associated with the exponent (the absence of a sign is assumed to be +). The absolute value of real data must either be zero, or between 1.0E-32 and 1.0E+32.
- A nonnumeric character in the numeric field (other than a valid convenience label, a parameter name, or an E or D exponent character) will be ignored and will cause the remainder of the field to be ignored. When the program can accept an alphanumeric label in a field, it will interpret any alphanumeric input that matches a valid label as the appropriate label. In other words, the program will not interpret a valid label as an identically named parameter. If you truly want to use a parameter in place of a valid label, you can use forced parametric substitution (using % signs). A non-alphanumeric character in a label field is taken as part of the label. Non-alphanumeric characters (such as CONTROL-characters, TAB-characters, and other terminal editing characters) should be avoided since they may be used directly instead of being interpreted by the terminal.

1.3.4. Condensed Data Input

An option for condensing the data input is available with the $ character. This character effectively ends a command and allows the next command to begin on the same line (at the next column). The line must not extend beyond 640 characters (including all commas and $ signs). If a command cannot be completed on the line, it should begin on the next line. Blank commands are ignored. The $ character should not be used following any command that causes a file switch to read additional commands (such as the **/INPUT**, ***USE**, **OPEXE**, **CDREAD**, etc. commands, any "unknown command" macro, or any do-loops or if-then-else constructs). The $ character should not be used between a command and its required following format line (such as ***VREAD**, ***VWRITE**, ***MSG**, etc.). The $ character should not be used after the **ALLSEL** command. Using the $ character in interactive mode may result in unexpected output behavior.

1.3.5. Units

The ANSYS program permits the use of any consistent set of units for length, force, time, temperature, etc. Care must be taken to ensure that all input data are in the same set of units. The **/UNITS** command may be used to note the system of units being used. Temperatures may be absolute or relative in most cases. For problems requiring absolute temperatures, such as those involving creep, swelling, or radiation, temperatures may be input as Centigrade or Fahrenheit, for convenience, with a zero temperature shift [**TOFFST**].

1.3.6. Defaults

ANSYS commands are of two types:

- specification commands
- action commands

Specifications define how an action is to be performed. If a specification is not defined before the action, the default specification is used. If some specifications are changed after the action, these changed specifications (and the remaining previous specifications, if any) will be used for the next action. If the same specification is defined more than once before the action, the last specification is used. Note, since specification settings are easily forgotten, reset the specification to the desired value before the action command.

To minimize the data input requirements, ANSYS commands operate on a "default" principle. That is, unless otherwise stated, default specifications are used whenever needed. Two types of default are used:

- the command default
- the argument default

The command default is defined as the action taken, or the argument values used, if the entire command is omitted. For action commands, the command default is "no action" unless otherwise stated. A description of the command default, when applicable, appears at the end of the Argument section of the command description. The argument default is the value the argument takes on if the command is included, but the argument on the command is left blank. The latter case is often used when only some arguments on a command are to be specified or to restore a default specification after it has been changed. The argument default is documented as part of each argument's description.

A default specification is invoked whenever no value (or a blank field) is input for the argument. If the argument accepts a numeric value and no default is specified, a blank field defaults to a zero value. Where a default is specified, a blank *or a zero value* will produce the default value (unless otherwise specified). If a zero value is desired where a zero input produces a nonzero default, input a small number (such as 1E-10) instead of zero. Specifications are initialized to their default values. Defaults which are not obvious are described, for the most part, with each command. For cases where defaults are not obvious and are not described, enter the desired value. Such cases may occur where no default is favored or where a default is purposely not documented (so that it may change in the future).

1.3.7. File Names

Various files are used during an ANSYS session for reading, writing, and storing data. ANSYS-generated files are identified by a name, a system-dependent separator, and an extension (as in the form **fname.ext**).

The name is obtained from the **Jobname** (250 characters maximum, including the directory name) as defined with the ANSYS execution command.

Note — Windows cannot create directory names longer than 245 characters.

Unless otherwise specified (see the *ANSYS Operations Guide*), the **Jobname** defaults to a system dependent name (usually **FILE**). The extension is a 2 to 4-character identifier (see Files that ANSYS Writes in the *ANSYS Basic Analysis Guide*). For example, the ANSYS error message file (which has the identifier **ERR**) with a Jobname of **FILE** and separator (.) would be generated as **FILE.ERR**. If the Jobname is **JOB1**, the file would be named **JOB1.ERR**. ANSYS documentation generically refers to ANSYS files as **Jobname**.identifier, such as **Jobname.ERR** for the error message file. Note, the extension may be shown upper case in the documentation but may actually be lower case on some operating systems (such as UNIX). Unless otherwise indicated, ANSYS files are written in the current working directory.

Some commands (such as **/INPUT** and **/OUTPUT**) allow you to specify a file name other than **Jobname**, an extension other than the default extension. Such user-written files use a naming convention similar to the ANSYS-generated files except that the name (248 characters maximum, including the directory name) is user defined, and the extension (8 characters maximum) is optional. The file name can include the directory path, but it is not necessary if you want to use the default directory. File names containing directory paths must reference existing directories--the program cannot create a directory. File and directory names may contain blank spaces, but strings containing blank spaces must be enclosed in single quotes.

A system dependent separator is automatically written between file name and extension (if the extension exists) input on commands.

File name fields are usually restricted to 248 characters on ANSYS commands, including the directory name. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

To help ensure portability of input files across systems, the ANSYS program has certain requirements for file names. File names specified by the user should be both valid per ANSYS criteria, and valid per the operating system. Avoid using special characters (+,-,*,/,$,!, etc.) in file names unless meaningful, and begin file names with a letter. Some system-dependent special characters (such as ~) are not interpreted the same as they are at the operating system level when used in pathnames. Parameter substitution may be forced within the name, extension, or directory fields by enclosing the parameter within percent (%) signs. For example, if I=5, the name TEST%I% becomes TEST5. Only one forced substitution may be made per field.

1.3.8. Star and Slash Commands

The slash (**/**) and star (*****) commands are usually used for supplying general control instructions to the ANSYS program. The slash commands are used, for example, for entering a processor (**/PREP7**, **/POST1**, etc.). Printout controls, display controls, and file controls are available within the slash command set. Repeat, looping and macro controls are available within the star command set. Many star and slash commands are global and apply to all processors. For example, the **/TITLE** command may be used to change the title in any processor (PREP7, POST1, etc.). The star commands can be input anywhere. Most slash commands can be input anywhere, however, some can only be input at a Begin level. The latter commands are specifically noted in the command explanation section. Graphics displays are controlled with the graphics display slash commands. The graphics display slash commands can be input anywhere.

Chapter 2: Command Groupings

This chapter contains tables of related commands. Each command in Chapter 3, “Command Dictionary” references one or more of these tables, which can then be consulted to see what other related commands are available. A brief description of each command is given.

The tables in this chapter provide helpful groupings of related commands. For example, if you want to define material types in the PREP7 general preprocessor, you can turn to Section 2.5: PREP7 Commands and see what commands are available under the heading "Table 2.26: “Materials”."

If you are viewing this manual in the ANSYS Help System, you can just click on the command name to display its description in Chapter 3, “Command Dictionary”.

You should not rely on this chapter as a source of procedural information but first look to the appropriate analysis guide to learn which commands are available for a particular type of analysis or operation. These guides contain guidelines on which commands to use for each step of the analysis.

The command summary tables are as follows:

Command Grouping Table Title
Section 2.1: SESSION Commands
Section 2.2: DATABASE Commands
Section 2.3: GRAPHICS Commands
Section 2.4: APDL Commands
Section 2.5: PREP7 Commands
Section 2.6: SOLUTION Commands
Section 2.7: POST1 Commands
Section 2.8: POST26 Commands
Section 2.9: AUX2 Commands
Section 2.10: AUX3 Commands
Section 2.11: AUX12 Commands
Section 2.12: AUX15 Commands
Section 2.13: RUNSTATS Commands
Section 2.14: OPTIMIZATION Commands
Section 2.15: VARIATIONAL TECHNOLOGY Commands
Section 2.16: PROBABILISTIC Design Commands
Section 2.17: DISPLAY Program Commands
Section 2.18: REDUCED Order Modeling Commands

2.1. SESSION Commands

These commands provide general control to the ANSYS session. The commands are grouped by functionality.

Table 2.1 Run Controls

These SESSION commands control the overall characteristics of the ANSYS session, including the jobname, Graphical User Interface behavior, and file switching.

/BATCH	Sets the program mode to "batch."
/CONFIG	Assigns values to ANSYS configuration parameters.
/CWD	Changes the current working directory.
/EOF	Exits the file being read.
/EXIT	Stops the run and returns control to the system.
/FILNAME	Changes the jobname for the analysis.
HELP	Displays help information on ANSYS commands and element types.
/INPUT	Switches the input file for the commands that follow.
KEYW	Sets a keyword used by the GUI for context filtering (GUI).
MEMM	Allows the current session to keep allocated memory
/MENU	Activates the Graphical User Interface (GUI).
/MSTART	Controls the initial GUI components.
/NERR	Limits the number of warning and error messages displayed.
/OUTPUT	Redirects text output to a file or to the screen.
/STATUS	Lists the status of items for the run.
/SYP	Passes a command string and arguments to the operating system.
/SYS	Passes a command string to the operating system.
/UI	Activates specified GUI dialog boxes.
/UIS	Controls the GUI behavior.

Table 2.2 Processor Entry

These SESSION commands are used to enter and exit the various processors in the program.

/AUX2	Enters the binary file dumping processor.
/AUX3	Enters the results file editing processor.
/AUX12	Enters the radiation matrix generation processor.
/AUX15	Enters the IGES file transfer processor.
FINISH	Exits normally from a processor.
/OPT	Enters the design optimizer.
/POST1	Enters the database results postprocessor.
/POST26	Enters the time-history results postprocessor.
/PREP7	Enters the model creation preprocessor.
/QUIT	Exits a processor.
/RUNST	Enters the run statistics processor.
/SOLU	Enters the solution processor.

Table 2.3 Files

These SESSION commands are for file operations, such as deleting, copying, and listing.

/ASSIGN	Reassigns a file name to an ANSYS file identifier.
/CLOG	Copies the session log file to a named file.
/COPY	Copies a file.
/DELETE	Deletes a file.
/FDELE	Deletes a binary file after it is used.
/FTYPE	Assigns an external or internal type to a binary file.
LGWRITE	Writes the database command log to a file.
***LIST**	Displays the contents of an external, coded file.
/RENAME	Renames a file.

Table 2.4 List Controls

These SESSION commands are used to control listings and printed program output.

C* **	Places a comment in the output.
/COM	Places a comment in the output.
/GO	Reactivates suppressed printout.
/GOLIST	Reactivates the suppressed data input listing.
/GOPR	Reactivates suppressed printout.
/NOLIST	Suppresses the data input listing.
/NOPR	Suppresses the expanded interpreted input data listing.

2.2. DATABASE Commands

These commands are used to operate on the database in a global sense. The commands are grouped by functionality.

Table 2.5 Set Up

These DATABASE commands can be used to initialize the database, save it to a file, or annotate it with titles and systems of units.

/CLEAR	Clears the database.
RESUME	Resumes the database from the database file.
SAVE	Saves all current database information.
/SMBC	Controls the display of solid model boundary condition symbols and labels.
STAT	Displays the status of database settings.
/STITLE	Defines subtitles.
/TITLE	Defines a main title.
UNDO	Allows the user to modify or save commands issued since the last **RESUME** or **SAVE** command.
/UNITS	Annotates the database with the system of units used.

Table 2.6 Selecting

These DATABASE commands are used to select subsets of database entities for further operations.

ALLSEL	Selects all entities with a single command.
ASLL	Selects those areas containing the selected lines.
ASEL	Selects a subset of areas.
ASLV	Selects those areas contained in the selected volumes.
DOFSEL	Selects a DOF label set for reference by other commands.
ESEL	Selects a subset of elements.
ESLA	Selects those elements associated with the selected areas.
ESLL	Selects those elements associated with the selected lines.
ESLN	Selects those elements attached to the selected nodes.
ESLV	Selects elements associated with the selected volumes.
KSEL	Selects a subset of keypoints or hard points.
KSLL	Selects those keypoints contained in the selected lines.
KSLN	Selects those keypoints associated with the selected nodes.
LSEL	Selects a subset of lines.
LSLA	Selects those lines contained in the selected areas.
LSLK	Selects those lines containing the selected keypoints.
NSEL	Selects a subset of nodes.
NSLA	Selects those nodes associated with the selected areas.
NSLE	Selects those nodes attached to the selected elements.
NSLK	Selects those nodes associated with the selected keypoints.
NSLL	Selects those nodes associated with the selected lines.
NSLV	Selects those nodes associated with the selected volumes.

These DATABASE commands are used to select subsets of database entities for further operations.

PARTSEL	Selects a subset of parts in an explicit dynamic analysis.
VSEL	Selects a subset of volumes.
VSLA	Selects those volumes containing the selected areas.

Table 2.7 Components

These DATABASE commands allow selected subsets of entities to be named as components for easy selection later on.

CM	Groups geometry items into a component.
CMDELE	Deletes a component or assembly definition.
CMEDIT	Edits an existing component or assembly.
CMGRP	Groups components and assemblies into an assembly.
CMLIST	Lists the entities contained in a component or assembly.
CMMOD	
CMPLOT	Plots the entities contained in a component or assembly.
CMSEL	Selects a subset of components and assemblies.

Table 2.8 Working Plane

These DATABASE commands turn on, move, rotate, and modify the working plane, which is used for picking operations.

KWPAVE	Moves the working plane origin to the average location of keypoints.
KWPLAN	Defines the working plane using three keypoints.
LWPLAN	Defines the working plane normal to a location on a line.
NWPAVE	Moves the working plane origin to the average location of nodes.
NWPLAN	Defines the working plane using three nodes.
WPAVE	Moves the working plane origin to the average of specified points.
WPCSYS	Defines the working plane location based on a coordinate system.
WPLANE	Defines a working plane to assist in picking operations.
WPOFFS	Offsets the working plane.
WPROTA	Rotates the working plane.
WPSTYL	Controls the display and style of the working plane.

Table 2.9 Coordinate System

These DATABASE commands define and manipulate coordinate systems.

CLOCAL	Defines a local coordinate system relative to the active coordinate system.
CS	Defines a local coordinate system by three node locations.
CSCIR	Locates the singularity for non-Cartesian local coordinate systems.
CSDELE	Deletes local coordinate systems.
CSKP	Defines a local coordinate system by three keypoint locations.
CSLIST	Lists coordinate systems.
CSWPLA	Defines a local coordinate system at the origin of the working plane.
CSYS	Activates a previously defined coordinate system.

These DATABASE commands define and manipulate coordinate systems.

LOCAL	Defines a local coordinate system by a location and orientation.

Table 2.10 Picking

These DATABASE commands are generated by the GUI when picking operations are performed.

FITEM	Identifies items chosen by a picking operation (GUI).
FLST	Specifies data required for a picking operation (GUI).

2.3. GRAPHICS Commands

These commands are used to control the graphics of the ANSYS program. The commands are grouped by functionality.

Table 2.11 Set Up

These GRAPHICS commands are used for general graphics control, such as window set up, color settings, specifications saving, etc.

/CMAP	Changes an existing or creates a new color mapping table.
/COLOR	Specifies the color mapping for various items.
/DEVICE	Controls graphics device options.
DSYS	Activates a display coordinate system for geometry listings and plots.
/DV3D	Sets 3-D device option modes.
/ERASE	Specifies that the screen is to be erased before each display.
ERASE	Explicitly erases the current display.
/GCMD	Controls the type of element or graph display used for the **GPLOT** command.
/GCOLUMN	Allows the user to apply a label to a specified curve.
/GFILE	Specifies the pixel resolution on Z-buffered graphics files.
GPLOT	Controls general plotting.
/GRAPHICS	Defines type of graphics display to be used for element plots.
/GRESUME	Sets graphics settings to the settings on a file.
/GSAVE	Saves graphics settings to a file for later use.
/GTYPE	Controls the entities that the **GPLOT** command displays.
HPGL	Specifies various HP options.
/IMAGE	Allows graphics data to be captured and saved.
IMMED	Allows immediate display of a model as it is generated.
JPEG	Provides JPEG File Export for ANSYS Displays.
/MREP	Enables you to reissue the graphics command macro "name" during a replot or zoom operation.
/NOERASE	Prevents the screen erase between displays.
/PCOPY	Generates hard copies automatically (device dependent).
PNGR	Sets file options for PNGR graphics export for ANSYS displays.
PSCR	Specifies various PostScript options.
/PSTATUS	Displays the global or window display specifications.
/REPLOT	Automatically reissues the last display command for convenience.
/RESET	Resets display specifications to their initial defaults.
/SEG	Allows graphics data to be stored in the local terminal memory.
/SHOW	Specifies the device and other parameters for graphics displays.
TIFF	Provides TIFF file Export for ANSYS Displays.
/WINDOW	Defines the window size on the screen.

Table 2.12 Views

These GRAPHICS commands are used to control the view of the model.

/ANGLE	Rotates the display about an axis.
/AUTO	Resets the focus and distance specifications to "automatically calculated."
/DIST	Specifies the viewing distance for magnifications and perspective.
/FOCUS	Specifies the focus point (center of the window).
/USER	Conveniently resets **/FOCUS** and **/DIST** to USER.
/VCONE	Defines the view cone angle for perspective displays.
/VIEW	Defines the viewing direction for the display.
/VUP	Specifies the global Cartesian coordinate system reference orientation.
/XFRM	Controls the centroid or the axis of dynamic rotation.
/ZOOM	Zooms a region of a display window.

Table 2.13 Scaling

These GRAPHICS commands are used to scale various parts of the display.

/DSCALE	Sets the displacement multiplier for displacement displays.
/ICLWID	Scales the line width of circuit builder icons.
/ICSCALE	Scales the icon size for elements supported in the circuit builder.
/RATIO	Distorts the object geometry.
/SHRINK	Shrinks elements, lines, areas, and volumes for display clarity.
/SSCALE	Sets the contour multiplier for topographic displays.
/TXTRE	Applies textured appearance to selected items.
/VSCALE	Scales the length of displayed vectors.

Table 2.14 Style

These GRAPHICS commands are used to change the way a model is displayed.

/CPLANE	Specifies the cutting plane for section and capped displays.
/CTYPE	Specifies the type of contour display.
/EDGE	Displays only the "edges" of an object.
/ESHAPE	Displays elements with shapes determined from the real constants.
/FACET	Specifies the facet representation used to form solid model displays.
/GLINE	Specifies the element outline style.
/GMARKER	Specifies the curve marking style.
GMFACE	Specifies the facet representation used to form solid models.
/LIGHT	Specifies the light direction for the display window.
/NORMAL	Allows displaying area elements by top or bottom faces.
/SHADE	Defines the type of surface shading used with Z-buffering.
/TRLCY	Specifies the level of translucency.
/TYPE	Defines the type of display.

Table 2.15 Labeling

These GRAPHICS commands are used to add helpful labels and symbols to displays.

/CFORMAT	Controls the graphical display of alpha character strings for parameters, components, assemblies, and tables.
/CLABEL	Specifies contour labeling.
/CONTOUR	Specifies the uniform contour values on stress displays.
/CVAL	Specifies nonuniform contour values on stress displays.
/GFORMAT	Specifies the format for the graphical display of numbers.
/HBC	Determines how boundary conditions are displayed in a display window.
/NUMBER	Specifies whether numbers, colors, or both are used for displays.
/PBC	Shows boundary condition symbols and values on displays.
/PBF	Shows body force loads as contours on displays.
PGSELE	Select a subset of elements for display with the PGR viewer.
/PICE	Shows initial conditions on elements as contours on displays.
/PLOPTS	Controls graphics options on subsequent displays.
/PNUM	Controls entity numbering/coloring on plots.
/PSF	Shows surface load symbols on model displays.
/PSYMB	Shows various symbols on displays.
/TRIAD	Shows the global XYZ coordinate triad on displays.

Table 2.16 Graphs

These GRAPHICS commands are used to control the way line graphs are displayed.

/AXLAB	Labels the X and Y axes on graph displays.
/GRID	Selects the type of grid on graph displays.
/GROPT	Sets various line graph display options.
/GRTYP	Selects single or multiple Y-axes graph displays.
/GTHK	Sets line thicknesses for graph lines.
/XRANGE	Specifies a linear abscissa (X) scale range.
/YRANGE	Specifies a linear ordinate (Y) scale range.

Table 2.17 Annotation

These GRAPHICS commands are used to annotate a display with notes and symbols.

/AN3D	Specifies 3-D annotation functions
/ANNOT	Activates graphics for annotating displays (GUI).
/ANUM	Specifies the annotation number, type, and hot spot (GUI).
/LARC	Creates annotation arcs (GUI).
/LINE	Creates annotation lines (GUI).
/LSPEC	Specifies annotation line attributes (GUI).
/LSYMBOL	Creates annotation symbols (GUI).
/PCIRCLE	Creates an annotation circle (GUI).
/PMORE	Creates an annotation polygon (GUI).
/POLYGON	Creates annotation polygons (GUI).

These GRAPHICS commands are used to annotate a display with notes and symbols.

/PSPEC	Creates annotation polygon attributes (GUI).
/PWEDGE	Creates an annotation wedge (GUI).
/TLABEL	Creates annotation text (GUI).
/TSPEC	Creates annotation text attributes (GUI).

2.4. APDL Commands

These commands make up the ANSYS Parametric Design Language (APDL). The commands are grouped by functionality.

Table 2.18 Parameter Definition

These APDL commands are used to define parameters and their values.

***AFUN**	Specifies units for angular functions in parameter expressions.
***ASK**	Prompts the user to input a parameter value.
***DEL**	Deletes a parameter (GUI).
***DIM**	Defines an array parameter and its dimensions.
***GET**	Retrieves a value and stores it as a user-named parameter.
/INQUIRE	Returns system information to a parameter.
PARRES	Reads parameters from a file.
PARSAV	Writes parameters to a file.
***SET**	Assigns values to user-named parameters.
***STATUS**	Lists the current parameters and abbreviations.
***TAXIS**	Defines table index numbers.
***TREAD**	Reads data from an external file into a table array parameter.
***VFILL**	Fills an array parameter.
***VGET**	Retrieves values into an array parameter.
***VREAD**	Reads data and produces an array parameter vector or matrix.

Table 2.19 Macro Files

These APDL commands are used to build and execute command macros.

***CFCLOS**	Closes the "command" file.
***CFOPEN**	Opens a "command" file.
***CFWRITE**	Writes an ANSYS command (or similar string) to a "command" file.
***CREATE**	Opens (creates) a macro file.
/DFLAB	Changes DOF labels for user custom elements.
***END**	Closes a macro file.
/MAIL	Mails file to specifed address.
***MSG**	Writes an output message via the ANSYS message subroutine.
/PMACRO	Specifies that macro contents be written to the session log file.
/PSEARCH	Specifies a directory to be searched for "unknown command" macro files.
/TEE	Writes a list of commands to a specified file at the same time that the commands are being executed.
***ULIB**	Identifies a macro library file.
***USE**	Executes a macro file.

Table 2.20 Abbreviations

These APDL commands can be used to define abbreviations for longer commands, and to create user-defined commands.

***ABBR**	Defines an abbreviation.
ABBRES	Reads abbreviations from a coded file.
ABBSAV	Writes the current abbreviation set to a coded file.
/UCMD	Assigns a user-defined command name.

Table 2.21 Array Parameters

These APDL commands are used to operate on parameter arrays (i.e., vectors and matrices).

/DIRECTORY	Put the file names in the current directory into a string parameter array.
***MFOURI**	Calculates the coefficients for, or evaluates, a Fourier series.
***MFUN**	Copies or transposes an array parameter matrix.
***MOPER**	Performs matrix operations on array parameter matrices.
***MWRITE**	Writes a matrix to a file in a formatted sequence.
***SREAD**	Reads a file into a string array parameter.
***TOPER**	Operates on table parameters.
***VABS**	Applies the absolute value function to array parameters.
***VCOL**	Specifies the number of columns in matrix operations.
***VCUM**	Allows array parameter results to add to existing results.
***VEDIT**	Allows numerical array parameters to be graphically edited.
***VFACT**	Applies a scale factor to array parameters.
***VFUN**	Performs a function on a single array parameter.
***VITRP**	Forms an array parameter by interpolation of a table.
***VLEN**	Specifies the number of rows to be used in array parameter operations.
***VMASK**	Specifies an array parameter as a masking vector.
***VOPER**	Operates on two array parameters.
***VPLOT**	Graphs columns (vectors) of array parameters.
***VPUT**	Restores array parameter values into the ANSYS database.
***VSCFUN**	Determines properties of an array parameter.
***VSTAT**	Lists the current specifications for the array parameters.
***VWRITE**	Writes data to a file in a formatted sequence.

Table 2.22 Process Controls

These APDL commands can be used to control the order in which other commands are processed.

***CYCLE**	Bypasses commands within a do-loop.
***DO**	Defines the beginning of a do-loop.
***ELSE**	Separates the final if-then-else block.
***ELSEIF**	Separates an intermediate if-then-else block.
***ENDDO**	Ends a do-loop and starts the looping action.
***ENDIF**	Ends an if-then-else.
***EXIT**	Exits a do-loop.

These APDL commands can be used to control the order in which other commands are processed.

***GO**	Causes a specified line on the input file to be read next.
***IF**	Conditionally causes commands to be read.
***REPEAT**	Repeats the previous command.
***RETURN**	Returns input stream to a higher level.
/WAIT	Causes a delay before the reading of the next command.

2.5. PREP7 Commands

These commands are used to create and set up the model. The commands are grouped by functionality.

Table 2.23 Database

These PREP7 commands are used to read model data into the database, list out the database, and control the numbering of entities in the database.

AFLIST	Lists the current data in the database.
CDREAD	Reads a file of solid model and database information into the database.
CDWRITE	Writes geometry and load database items to a file.
CDOPT	Specifies format to be used for archiving geometry.
CECHECK	Check constraint equations and couplings for rigid body motions.
CHECK	Checks current database items for completeness.
CNCHECK	Lists the initial status of contact pairs.
FC	Provides failure criteria information and activates a data table to input temperature-dependent stress and strain limits.
FCCHECK	Checks both the strain and stress input criteria for all materials.
FCDELE	Deletes previously defined failure criterion data for the given material.
FCLIST	To list what the failure criteria is that you have input.
IGESOUT	Writes solid model data to a file in IGES Version 5.1 format.
NOOFFSET	Prevents the **CDREAD** command from offsetting specified data items.
NUMCMP	Compresses the numbering of defined items.
NUMMRG	Merges coincident or equivalently defined items.
NUMOFF	Adds a number offset to defined items.
NUMSTR	Establishes starting numbers for automatically numbered items.
/PREP7	Enters the model creation preprocessor.

Table 2.24 Element Type

These PREP7 commands define the type of elements to be used in the model.

DOF	Adds degrees of freedom to the current DOF set.
ET	Defines a local element type from the element library.
ETCHG	Changes element types to their corresponding types.
ETCONTROL	Controls the element technologies used in element formulation (for applicable elements).
ETDELE	Deletes element types.
ETLIST	Lists currently defined element types.
KEYOPT	Sets element key options.
NSVR	Defines the number of variables for user-programmable element options.

Table 2.25 Real Constants

These PREP7 commands define the model real constants.

R	Defines the element real constants.
RDELE	Deletes real constant sets.

These PREP7 commands define the model real constants.

RLIST	Lists the real constant sets.
RMODIF	Modifies real constant sets.
RMORE	Adds real constants to a set.
SETFGAP	Updates real constant table for squeeze film elements.

Table 2.26 Materials

These PREP7 commands are used to define the linear material properties.

EMUNIT	Specifies the system of units for magnetic field problems.
***EVAL**	Evaluates hyperelastic constants.
***MOONEY**	Calculates Mooney-Rivlin hyperelastic constants from test data.
MP	Defines a linear material property.
MPAMOD	Modifies temperature-dependent coefficients of thermal expansion.
MPCHG	Changes the material number attribute of an element.
MPCOPY	Copies material model data from one material reference number to another.
MPDATA	Defines property data to be associated with the temperature table.
MPDELE	Deletes linear material properties.
MPDRES	Reassembles existing material data with the temperature table.
/MPLIB	Sets the default material library read and write paths.
MPLIST	Lists linear material properties.
MPPLOT	Plots linear material properties as a function of temperature.
MPREAD	Reads a file containing material properties.
MPTEMP	Defines a temperature table for material properties.>
MPTGEN	Adds temperatures to the temperature table by generation.
MPTRES	Restores a temperature table previously defined.
MPWRITE	Writes linear material properties in the database to a file (if the Lib option is not specified) or writes both linear and nonlinear material properties (if Lib is specified) from the database to a file.
TBFT	Performs curve-fitting operations on materials.
UIMP	Defines constant material properties (GUI).

Table 2.27 Data Tables

These PREP7 commands create and modify the data tables, used to define nonlinear materials, for example.

TB	Activates a data table.
TBCOPY	Copies a data table from one material to another.
TBDATA	Defines data for the data table.
TBDELE	Deletes previously defined data tables.
TBLIST	Lists the data tables.
TBMODIF	Modifies data for the data table (GUI).
TBPLOT	Displays the data table.
TBPT	Defines a point on a stress-strain or B-H curve.
TBTEMP	Defines a temperature for the data table.

Table 2.28 Primitives

These PREP7 commands are used to create primitive shapes for modeling.

BLC4	Creates a rectangular area or block volume by corner points.
BLC5	Creates a rectangular area or block volume by center and corner points.
BLOCK	Creates a block volume based on working plane coordinates.
CON4	Creates a conical volume anywhere on the working plane.
CONE	Creates a conical volume centered about the working plane origin.
CYL4	Creates a circular area or cylindrical volume anywhere on the working plane.
CYL5	Creates a circular area or cylindrical volume by end points.
CYLIND	Creates a cylindrical volume centered about the working plane origin.
PCIRC	Creates a circular area centered about the working plane origin.
POLY	Creates a polygonal area based on working plane coordinate pairs.
PRI2	Creates a polygonal area or a prism volume by vertices (GUI).
PRISM	Creates a prism volume based on working plane coordinate pairs.
PTXY	Defines coordinate pairs for use in polygons and prisms.
RECTNG	Creates a rectangular area anywhere on the working plane.
RPOLY	Creates a regular polygonal area centered about the working plane origin.
RPR4	Creates a regular polygonal area or prism volume anywhere on the working plane.
RPRISM	Creates a regular prism volume centered about the working plane origin.
SPH4	Creates a spherical volume anywhere on the working plane.
SPH5	Creates a spherical volume by diameter end points.
SPHERE	Creates a spherical volume centered about the working plane origin.
TORUS	Creates a toroidal volume.

Table 2.29 Keypoints

These PREP7 commands are used to create, modify, list, etc., keypoints.

GSUM	Calculates and prints geometry items.
K	Defines a keypoint.
KBETW	Creates a keypoint between two existing keypoints.
KCENTER	Creates a keypoint at the center of a circular arc defined by three locations.
KDELE	Deletes unmeshed keypoints.
KDIST	Calculates and lists the distance between two keypoints.
KFILL	Generates keypoints between two keypoints.
KGEN	Generates additional keypoints from a pattern of keypoints.
KL	Generates a keypoint at a specified location on an existing line.
KLIST	Lists the defined keypoints or hard points.
KMODIF	Modifies an existing keypoint.
KMOVE	Calculates and moves a keypoint to an intersection.
KNODE	Defines a keypoint at an existing node location.
KPLOT	Displays the selected keypoints.
KPSCALE	Generates a scaled set of (meshed) keypoints from a pattern of keypoints.

These PREP7 commands are used to create, modify, list, etc., keypoints.

KSCALE	Generates a scaled pattern of keypoints from a given keypoint pattern.
KSUM	Calculates and prints geometry statistics of the selected keypoints.
KSYMM	Generates a reflected set of keypoints.
KTRAN	Transfers a pattern of keypoints to another coordinate system.
SOURCE	Defines a default location for undefined nodes or keypoints.

Table 2.30 Hard Points

These PREP7 commands are used to create, modify, list, etc., hard points.

HPTCREATE	Defines a hard point.
HPTDELETE	Deletes selected hard points.

Table 2.31 Lines

These PREP7 commands are used to create, modify, list, etc., lines.

BSPLIN	Generates a single line from a spline fit to a series of keypoints.
CIRCLE	Generates circular arc lines.
GSUM	Calculates and prints geometry items.
L	Defines a line between two keypoints.
L2ANG	Generates a line at an angle with two existing lines.
L2TAN	Generates a line tangent to two lines.
LANG	Generates a straight line at an angle with a line.
LARC	Defines a circular arc.
LAREA	Generates the shortest line between two keypoints on an area.
LCOMB	Combines adjacent lines into one line.
LDELE	Deletes unmeshed lines.
LDIV	Divides a single line into two or more lines.
LDRAG	Generates lines by sweeping a keypoint pattern along path.
LEXTND	Extends a line at one end by using its slope.
LFILLT	Generates a fillet line between two intersecting lines.
LGEN	Generates additional lines from a pattern of lines.
LLIST	Lists the defined lines.
LPLOT	Displays the selected lines.
LREVERSE	Reverses the normal of a line, regardless of its connectivity or mesh status.
LROTAT	Generates circular lines by rotating a keypoint pattern about an axis.
LSSCALE	Generates a scaled set of lines from a pattern of lines.
LSTR	Defines a straight line irrespective of the active coordinate system.
LSUM	Calculates and prints geometry statistics of the selected lines.
LSYMM	Generates lines from a line pattern by symmetry reflection.
LTAN	Generates a line at the end of, and tangent to, an existing line.
LTRAN	Transfers a pattern of lines to another coordinate system.
SPLINE	Generates a segmented spline through a series of keypoints.
SSLN	Selects and displays small lines in the model.

Table 2.32 Areas

These PREP7 commands are used to create, modify, list, etc., areas.

A	Defines an area by connecting keypoints.
AATT	Associates element attributes with the selected, unmeshed areas.
ADELE	Deletes unmeshed areas.
ADGL	Lists keypoints of an area that lie on a parametric degeneracy.
ADRAG	Generates areas by dragging a line pattern along a path.
AFILLT	Generates a fillet at the intersection of two areas.
AGEN	Generates additional areas from a pattern of areas.
AL	Generates an area bounded by previously defined lines.
ALIST	Lists the defined areas.
ANORM	Reorients area normals.
AOFFST	Generates an area, offset from a given area.
APLOT	Displays the selected areas.
AREVERSE	Reverses the normal of an area, regardless of its connectivity or mesh status.
AROTAT	Generates cylindrical areas by rotating a line pattern about an axis.
ARSCALE	Generates a scaled set of areas from a pattern of areas.
ARSYM	Generates areas from an area pattern by symmetry reflection.
ASKIN	Generates an area by "skinning" a surface through guiding lines.
ASUB	Generates an area using the shape of an existing area.
ASUM	Calculates and prints geometry statistics of the selected areas.
ATRAN	Transfers a pattern of areas to another coordinate system.
GSUM	Calculates and prints geometry items.
SPLOT	Displays the selected areas and a faceted view of their underlying surfaces

Table 2.33 Volumes

These PREP7 commands are used to create, modify, list, etc., volumes.

EXTOPT	Controls options relating to the generation of volume elements from area elements.
GSUM	Calculates and prints geometry items.
V	Defines a volume through keypoints.
VA	Generates a volume bounded by existing areas.
VDELE	Deletes unmeshed volumes.
VDGL	Lists keypoints of a volume that lie on a parametric degeneracy.
VDRAG	Generates volumes by dragging an area pattern along a path.
VEXT	Generates additional volumes by extruding areas.
VGEN	Generates additional volumes from a pattern of volumes.
VLIST	Lists the defined volumes.
VLSCALE	Generates a scaled set of volumes from a pattern of volumes.
VOFFST	Generates a volume, offset from a given area.
VPLOT	Displays the selected volumes.
VROTAT	Generates cylindrical volumes by rotating an area pattern about an axis.

These PREP7 commands are used to create, modify, list, etc., volumes.

VSUM	Calculates and prints geometry statistics of the selected volumes.
VSYMM	Generates volumes from a volume pattern by symmetry reflection.
VTRAN	Transfers a pattern of volumes to another coordinate system.

Table 2.34 Booleans

These PREP7 commands are used to perform Boolean operations on solid model entities.

AADD	Adds separate areas to create a single area.
AGLUE	Generates new areas by "gluing" areas.
AINA	Finds the intersection of areas.
AINP	Finds the pairwise intersection of areas.
AINV	Finds the intersection of an area with a volume.
AOVLAP	Overlaps areas.
APTN	Partitions areas.
ASBA	Subtracts areas from areas.
ASBL	Subtracts lines from areas.
ASBV	Subtracts volumes from areas.
ASBW	Subtracts the intersection of the working plane from areas (divides areas).
BOPTN	Specifies Boolean operation options.
BTOL	Specifies the Boolean operation tolerances.
LCSL	Divides intersecting lines at their point(s) of intersection.
LGLUE	Generates new lines by "gluing" lines.
LINA	Finds the intersection of a line with an area.
LINL	Finds the common intersection of lines.
LINP	Finds the pairwise intersection of lines.
LINV	Finds the intersection of a line with a volume.
LOVLAP	Overlaps lines.
LPTN	Partitions lines.
LSBA	Subtracts areas from lines.
LSBL	Subtracts lines from lines.
LSBV	Subtracts volumes from lines.
LSBW	Subtracts the intersection of the working plane from lines (divides lines).
VADD	Adds separate volumes to create a single volume.
VGLUE	Generates new volumes by "gluing" volumes.
VINP	Finds the pairwise intersection of volumes.
VINV	Finds the intersection of volumes.
VOVLAP	Overlaps volumes.
VPTN	Partitions volumes.
VSBA	Subtracts areas from volumes.
VSBV	Subtracts volumes from volumes.
VSBW	Subtracts intersection of the working plane from volumes (divides volumes).

Table 2.35 Meshing

These PREP7 commands are used to mesh solid models with nodes and elements.

ACCAT	Concatenates multiple areas in preparation for mapped meshing.
ACLEAR	Deletes nodes and area elements associated with selected areas.
AESIZE	Specifies the element size to be meshed onto areas.
AMAP	Generates a 2-D mapped mesh based on specified area corners.
AMESH	Generates nodes and area elements within areas.
AREFINE	Refines the mesh around specified areas.
CHKMSH	Checks area and volume entities for previous meshes.
CLRMSHLN	Clears meshed entities.
CPCYC	Couples the two side faces of a cyclically symmetric model for loadings that are the same on every segment.
DESIZE	Controls default element sizes.
EORIENT	Reorients solid element normals.
EREFINE	Refines the mesh around specified elements.
ESIZE	Specifies the default number of line divisions.
ESYS	Sets the element coordinate system attribute pointer.
FVMESH	Generates nodes and tetrahedral volume elements from detached exterior area elements (facets).
GSGDATA	Specifies the reference point and defines the geometry in the fiber direction for the generalized plane strain element option.
IMESH	Generates nodes and interface elements along lines or areas.
KATT	Associates attributes with the selected, unmeshed keypoints.
KCLEAR	Deletes nodes and point elements associated with selected keypoints.
KESIZE	Specifies the edge lengths of the elements nearest a keypoint.
KMESH	Generates nodes and point elements at keypoints.
KREFINE	Refines the mesh around specified keypoints.
KSCON	Specifies a keypoint about which an area mesh will be skewed.
LATT	Associates element attributes with the selected, unmeshed lines.
LCCAT	Concatenates multiple lines into one line for mapped meshing.
LCLEAR	Deletes nodes and line elements associated with selected lines.
LESIZE	Specifies the divisions and spacing ratio on unmeshed lines.
LMESH	Generates nodes and line elements along lines.
LREFINE	Refines the mesh around specified lines.
MAT	Sets the element material attribute pointer.
MCHECK	Checks mesh connectivity.
MODMSH	Controls the relationship of the solid model and the FE model.
MOPT	Specifies meshing options.
MSHAPE	For elements that support multiple shapes, specifies the element shape to be used for meshing.
MSHCOPY	Simplifies the generation of meshes that have matching node element patterns on two different line groups (in 2-D) or area groups (3-D).
MSHKEY	Specifies whether free meshing or mapped meshing should be used to mesh a model.

These PREP7 commands are used to mesh solid models with nodes and elements.

MSHMID	Specifies placement of midside nodes.
MSHPATTERN	Specifies pattern to be used for mapped triangle meshing.
NREFINE	Refines the mesh around specified nodes.
PSMESH	Splits an initially continuous group of beam, shell, plane, or solid elements into two unconnected groups, tied together with PRETS179 pretension elements.
REAL	Sets the element real constant set attribute pointer.
RTHICK	Defines variable thickness at nodes for shell elements.
SHPP	Controls element shape checking.
SMRTSIZE	Specifies meshing parameters for automatic (smart) element sizing.
TCHG	Converts 20-node degenerate tetrahedral elements to their 10-node non-degenerate counterparts.
TIMP	Improves the quality of tetrahedral elements that are not associated with a volume.
TYPE	Sets the element type attribute pointer.
VATT	Associates element attributes with the selected, unmeshed volumes.
VCLEAR	Deletes nodes and volume elements associated with selected volumes.
VIMP	Improves the quality of the tetrahedral elements in the selected volume(s).
VMESH	Generates nodes and volume elements within volumes.
VSWEEP	Fills an existing unmeshed volume with elements by sweeping the mesh from an adjacent area throughout the volume.

Table 2.36 Nodes

These PREP7 commands are used to create, modify, list, etc., nodes.

CENTER	Defines a node at the center of curvature of 2 or 3 nodes.
FILL	Generates a line of nodes between two existing nodes.
MOVE	Calculates and moves a node to an intersection.
N	Defines a node.
NANG	Rotates a nodal coordinate system by direction cosines.
NDELE	Deletes nodes.
NDIST	Calculates and lists the distance between two nodes.
NGEN	Generates additional nodes from a pattern of nodes.
NKPT	Defines a node at an existing keypoint location.
NLIST	Lists nodes.
NMODIF	Modifies an existing node.
NPLOT	Displays nodes.
NREAD	Reads nodes from a file.
NROTAT	Rotates nodal coordinate systems into the active system.
NRRANG	Specifies the range of nodes to be read from the node file.
NSCALE	Generates a scaled set of nodes from a pattern of nodes.
NSMOOTH	Smooths selected nodes among selected elements.
NSYM	Generates a reflected set of nodes.
NWRITE	Writes nodes to a file.

These PREP7 commands are used to create, modify, list, etc., nodes.

QUAD	Generates a quadratic line of nodes from three nodes.
SOURCE	Defines a default location for undefined nodes or keypoints.
TRANSFER	Transfers a pattern of nodes to another coordinate system.

Table 2.37 Elements

These PREP7 commands are used to create, modify, list, etc., elements.

AFSURF	Generates surface elements overlaid on the surface of existing solid elements and assigns the extra node as the closes fluid element node.
E	Defines an element by node connectivity.
EDELE	Deletes selected elements from the model.
EGEN	Generates elements from an existing pattern.
EINTF	Defines two-node elements between coincident nodes.
ELIST	Lists the elements.
EMID	Adds or removes midside nodes.
EMODIF	Modifies a previously defined element.
EMORE	Adds more nodes to the just-defined element.
EMTGEN	Generates a set of TRANS126 elements
EN	Defines an element by its number and node connectivity.
ENDRELEASE	Specifies degrees of freedom to be decoupled for end release.
ENGEN	Generates elements from an existing pattern.
ENORM	Reorients shell element normals.
ENSYM	Generates elements by symmetry reflection.
EORIENT	Reorients solid element normals.
EPLOT	Produces an element display.
EREAD	Reads elements from a file.
ERRANG	Specifies the element range to be read from a file.
ESURF	Generates elements overlaid on the free faces of existing elements.
ESYM	Generates elements from a pattern by a symmetry reflection.
ESYS	Sets the element coordinate system attribute pointer.
EWRITE	Writes elements to a file.
LAYLIST	Lists real constants material properties for layered elements.
LAYPLOT	Displays the layer stacking sequence for layered elements.
LFSURF	Generates surface elements overlaid on the edge of existing solid elements and assigns the extra node as the closest fluid element node.
MAT	Sets the element material attribute pointer.
NDSURF	Generates surface elements overlaid on the edge of existing elements and assigns the extra node as the closest fluid element node.
REAL	Sets the element real constant set attribute pointer.
SHSD	Creates or deletes shell-solid interface to be used in shell-to-solid assemblies
TSHAP	Defines simple 2-D and 3-D geometric surfaces for target segment elements.
TYPE	Sets the element type attribute pointer.

These PREP7 commands are used to create, modify, list, etc., elements.

UPGEOM	Adds displacements from a previous analysis and updates the geometry to the deformed configuration.

Table 2.38 Superelements

These PREP7 commands are used to create and modify superelements.

SE	Defines a superelement.
SEDLIST	Lists the DOF solution of a superelement after the use pass.
SELIST	Lists the contents of a superelement matrix file.
SESYMM	Performs a symmetry operation on a superelement within the use pass.
SETRAN	Creates a superelement from an existing superelement.

Table 2.39 Piping

These PREP7 commands are used to conveniently create models of piping systems.

BELLOW	Defines a bellows in a piping run.
BEND	Defines a bend in a piping run.
BRANCH	Defines the starting point for a piping branch.
FLANGE	Defines a flange in a piping run.
MITER	Defines a mitered bend in a piping run.
PCORRO	Specifies the allowable exterior corrosion thickness for a piping run.
PDRAG	Defines the external fluid drag loading for a piping run.
PFLUID	Defines the contained fluid density for a piping run.
PGAP	Defines a spring-gap constraint in a piping run.
PINSUL	Defines the external insulation constants in a piping run.
POPT	Selects the piping analysis standard for a piping run.
PPRES	Defines the internal pressure for a piping run.
PSPEC	Defines pipe material and dimensions.
PSPRNG	Defines a spring constraint in a piping run.
PTEMP	Defines the pipe wall temperatures in a piping run.
PUNIT	Selects the system of length units to be used in a piping run.
REDUCE	Defines a reducer in a piping run.
RUN	Defines a pipe run.
TEE	Defines a tee in a piping run.
VALVE	Defines a valve in a piping run.

Table 2.40 Digitizing

These PREP7 commands are used to define nodes by tablet digitizing.

DIG	Digitizes nodes to a surface.
DMOVE	Digitizes nodes on surfaces and along intersections.
DSET	Sets the scale and drawing plane orientation for a digitizing tablet.
DSURF	Defines the surface upon which digitized nodes lie.

Table 2.41 Coupled DOF

These PREP7 commands are used to define, modify, list, etc., coupled degrees of freedom.

CP	Defines (or modifies) a set of coupled degrees of freedom.
CPDELE	Deletes coupled degree of freedom sets.
CPINTF	Defines coupled degrees of freedom at an interface.
CPLGEN	Generates sets of coupled nodes from an existing set.
CPLIST	Lists the coupled degree of freedom sets.
CPNGEN	Defines, modifies, or adds to a set of coupled degrees of freedom.
CPSGEN	Generates sets of coupled nodes from existing sets.

Table 2.42 Constraint Equations

These PREP7 commands are used to define, modify, list, etc., constraint equations.

CE	Defines a constraint equation relating degrees of freedom.
CECYC	Generates the constraint equations for a cyclic symmetry analysis.
CEDELE	Deletes constraint equations.
CEINTF	Generates constraint equations at an interface.
CELIST	Lists the constraint equations.
CERIG	Defines a rigid region.
CESGEN	Generates a set of constraint equations from existing sets.
RBE3	Distributes the force/moment applied at the master node to a set of slave nodes, taking into account the geometry of the slave nodes as well as weighting factors.

Table 2.43 Element Reordering

These PREP7 commands are used to reorder the model wavefront.

NOORDER	Re-establishes the original element ordering.
WAVES	Initiates reordering.
WERASE	Erases all reordering wave lists.
WFRONT	Estimates wavefront statistics.
WMID	Specifies reordering options for the **WAVES** command.
WMORE	Adds more nodes to the starting wave list.
WSORT	Initiates element reordering based upon a geometric sort.
WSTART	Defines a starting wave list.

Table 2.44 FLOTRAN Options

These PREP7 commands are used to specify the options and output controls for a FLOTRAN CFD analysis.

FLDATA	Sets up a FLOTRAN analysis.
FLDATA1	Controls which features of the solution algorithm are activated.
FLDATA2	Sets iteration and output controls for steady state analyses.
FLDATA3	Sets the convergence monitors for the degree of freedom set.

These PREP7 commands are used to specify the options and output controls for a FLOTRAN CFD analysis.

FLDATA4	Sets controls for transient analyses based on transient time and convergence monitors or sets time integration method.
FLDATA4A	Sets controls for transient analyses based on the number of time steps.
FLDATA5	Sets output and storage controls.
FLDATA6	Controls the output of the convergence monitor.

Table 2.45 FLOTRAN Property

These PREP7 commands are used to specify the fluid properties for a FLOTRAN CFD analysis.

FLDATA7	Specifies the type of fluid property.
FLDATA8	Specifies the NOMI coefficient of the fluid property equation.
FLDATA9	Specifies the COF1 coefficient of the fluid property equation.
FLDATA10	Specifies the COF2 coefficient of the fluid property equation.
FLDATA11	Specifies the COF3 coefficient of the fluid property equation.
FLDATA12	Sets the property update frequency flag.
FLDATA13	Sets the property variation flag.

Table 2.46 FLOTRAN Operating

These PREP7 commands are used to specify the operating conditions for a FLOTRAN CFD analysis.

FLDATA14	Specifies the reference temperature.
FLDATA15	Specifies the reference pressure.
FLDATA16	Specifies the bulk modulus parameter.
FLDATA17	Specifies the specific heat ratio.

Table 2.47 FLOTRAN Solver

These PREP7 commands are used to control the solver for a FLOTRAN CFD analysis.

FLDATA18	Selects the algebraic solver.
FLDATA19	Specifies the number of TDMA sweeps.
FLDATA20	Specifies the number of conjugate direction search vectors.
FLDATA20A	Specifies the amount of fill-in when preconditioning the coefficient matrix.
FLDATA20B	Specifies the number of fill-ins for the ILU preconditioner.
FLDATA21	Specifies the convergence monitor.
FLDATA22	Specifies the maximum number of semi-direct iterations.
FLDATA23	Specifies the solver minimum normalized rate of change.

Table 2.48 FLOTRAN Turbulence

These PREP7 commands are used to control the turbulence setting for a FLOTRAN CFD analysis.

FLDATA24	Sets the turbulence model and the constants used in the Standard k-ε Model and the Zero Equation Turbulence Model.
FLDATA24A	Sets constants for the Re-Normalized Group Turbulence Model (RNG).
FLDATA24B	Sets constants for the New k-ε Model due to Shih (NKE).

These PREP7 commands are used to control the turbulence setting for a FLOTRAN CFD analysis.

FLDATA24C	Sets constants for the Nonlinear Model of Girimaji (GIR).
FLDATA24D	Sets constants for the Shih, Zhu, Lumley Model (SZL).
FLDATA24E	Sets constants for the k-ω Model.
FLDATA24F	Sets the turbulent production clip factor for the Shear Stress Transport (SST) model.
FLDATA24G	Sets constants in the k-ω regime for the Shear Stress Transport (SST) model.
FLDATA24H	Sets constants in the k-ε regime for the Shear Stress Transport (SST) model.
FLDATA40	Controls activation of thermal stabilization near walls.

Table 2.49 FLOTRAN Stability

These PREP7 commands are used to control the stability settings for a FLOTRAN CFD analysis.

FLDATA25	Sets solution and property relaxation factors.
FLDATA26	Sets stability controls.
FLDATA34	Sets modified inertial relaxation factors.

Table 2.50 FLOTRAN Miscellaneous

These PREP7 commands are used to control miscellaneous settings for a FLOTRAN CFD analysis.

FLDATA27	Controls dependent variable printing.
FLDATA28	Specifies that variable results are to be replaced.
FLDATA29	Re-initializes a results variable.
FLDATA30	Controls the quadrature orders.
FLDATA31	Specifies dependent variable caps.
FLDATA32	Controls restart options.
FLDATA33	Specifies the approach to discretize the advection term.
FLDATA35	Specifies tolerances for the lower and upper bound of the volume fraction.
FLDATA36	Specifies ambient reference values outside of the fluid for the volume of fluid (VOF) method.
FLDATA37	Specifies the segregated solution algorithm.
FLDATA38	Specifies the mass type for a fluid transient analysis.
FLDATA39	Specifies remeshing parameters for transient fluid flow and fluid-solid interaction analyses.
ICVFRC	Sets the initial volume fraction field for a geometry.
PLVFRC	Displays volume fractions in a volume of fluid (VOF) analysis.

Table 2.51 FLOTRAN Multiple Species

These PREP7 commands are used for multiple species transport in a FLOTRAN CFD analysis.

MSADV	Specifies the approach to discretize the advection term in a species transport equation.
MSCAP	Activates and controls mass fraction capping for a species.
MSDATA	Defines multiple species data applicable to all species.
MSMASS	Specifies the mass type for a transient species analysis.
MSMETH	Specifies the method of solution of the species transport equations.

These PREP7 commands are used for multiple species transport in a FLOTRAN CFD analysis.

MSMIR	Sets modified inertial relaxation factors for multiple species.
MSNOMF	Specifies the initial value of nominal mass fraction for a species.
MSPROP	Defines the fluid properties of a species.
MSQUAD	Specifies the quadrature order for multiple species elements.
MSRELAX	Specifies relaxation factors for a multiple species transport analysis.
MSSOLU	Specifies solution options for multiple species transport.
MSSPEC	Specifies the name, molecular weight, and Schmidt number of a species.
MSTERM	Sets the convergence monitors for species.
MSVARY	Allows species properties to vary between global iterations.

Table 2.52 Special Purpose

These PREP7 commands are used for certain special purpose operations.

/CYCEXPAND	Verifies a cyclically symmetric model by graphically expanding it partially or into the full 360 degrees.
CYCLIC	Specifies a cyclic symmetry analysis.
CYCOPT	Specifies solution options for a cyclic symmetry analysis.
EMSYM	Specifies circular symmetry for electromagnetic sources.
HFEREFINE	Automatically refines high-frequency tetrahedral elements (HF119) or lists high-frequency brick elements (HF120) with the largest error.
PERBC2D	Generates periodic constraints for 2-D planar magnetic field analyses.
PHYSICS	Writes, reads, or lists all element information.
RACE	Defines a "racetrack" current source.

Table 2.53 Status

These PREP7 commands are for use with the STAT command.

AREAS	Specifies "Areas" as the subsequent status topic.
BOOL	Specifies "Booleans" as the subsequent status topic.
CEQN	Specifies "Constraint equations" as the subsequent status topic.
COUPLE	Specifies "Node coupling" as the subsequent status topic.
DIGIT	Specifies "Node digitizing" as the subsequent status topic.
ELEM	Specifies "Elements" as the subsequent status topic.
ETYPE	Specifies "Element types" as the subsequent status topic.
FATIGUE	Specifies "Fatigue data status" as the subsequent status topic.
FEBODY	Specifies "Body loads on elements" as the subsequent status topic.
FECONS	Specifies "Constraints on nodes" as the subsequent status topic.
FEFOR	Specifies "Forces on nodes" as the subsequent status topic.
FESURF	Specifies "Surface loads on elements" as the subsequent status topic.
FLOTRAN	Specifies "FLOTRAN data settings" as the subsequent status topic.
GEOMETRY	Specifies "Geometry" as the subsequent status topic.
KEYPTS	Specifies "Keypoints" as the subsequent status topic.
LINE	Specifies "Lines" as the subsequent status topic.
MATER	Specifies "Material properties" as the subsequent status topic.

These PREP7 commands are for use with the STAT command.

MESHING	Specifies "Meshing" as the subsequent status topic.
NODES	Specifies "Nodes" as the subsequent status topic.
PIPE	Specifies "Pipe modeling" as the subsequent status topic.
PMETH	Specifies "p-Method" as the subsequent status topic.
PRIM	Specifies "Solid model primitives" as the subsequent status topic.
RCON	Specifies "Real constants" as the subsequent status topic.
REORDER	Specifies "Model reordering" as the subsequent status topic.
SELM	Specifies "Superelements" as the subsequent status topic.
TBLE	Specifies "Data table properties" as the subsequent status topic.
VOLUMES	Specifies "Volumes" as the subsequent status topic.

Table 2.54 Explicit Dynamics

These PREP7 commands are used for an explicit dynamic analysis.

EDASMP	Creates a part assembly to be used in an explicit dynamic analysis.
EDBOUND	Defines a boundary plane for sliding or cyclic symmetry.
EDBX	Creates a box shaped volume to be used in a contact definition.
EDCGEN	Specifies contact parameters for an explicit dynamic analysis.
EDCLIST	Lists all contact entity specifications in an explicit dynamic analysis.
EDCMORE	Specifies additional contact parameters for a given contact definition in an explicit dynamic analysis.
EDCNSTR	Defines various types of constraints for an explicit dynamic analysis.
EDCONTACT	Specifies contact surface controls for an explicit dynamic analysis.
EDCRB	Constrains two rigid bodies to act as one in an explicit dynamic analysis.
EDCURVE	Specifies data curves for an explicit dynamic analysis.
EDDBL	Sets the single or double precision version of LS-DYNA into effect.
EDDC	Deletes or deactivates/reactivates contact entity specifications in an explicit dynamic analysis.
EDIPART	Defines inertia for rigid parts in an explicit dynamic analysis.
EDLCS	Defines a local coordinate system for use in explicit dynamic analysis.
EDMP	Defines material properties for an explicit dynamic analysis.
EDNB	Defines a nonreflecting boundary in an explicit dynamic analysis.
EDNDTSD	Allows smoothing of noisy data for explicit dynamic analyses and provides a graphical representation of the data.
EDNROT	Applies a rotated coordinate nodal constraint in an explicit dynamic analysis.
EDPART	Configures parts for an explicit dynamic analysis.
EDPC	Selects and plots explicit dynamic contact entities.
EDSP	Specifies small penetration checking for contact entities in an explicit dynamic analysis.
EDWELD	Defines a massless spotweld or generalized weld for use in an explicit dynamic analysis.

Table 2.55 CAD Repair

These PREP7 commands are for IGES and Connection import.

ALPFILL	Fills in an area loop within an existing 2-D area (for models imported from CAD files).
ARCOLLAPSE	Collapses specified area to a specified line segment (for models imported from CAD files).
ARDETACH	Detaches areas from neighboring geometrical entities (for models imported from CAD files).
ARFILL	Creates an area based on a set of singly-connected lines (for models imported from CAD files).
ARMERGE	Merges two or more singly-connected adjacent areas (for models imported from CAD files).
ARSPLIT	Splits an area between two keypoints (for models imported from CAD files).
GAPFINISH	Exits from the CAD import topology repair stage.
GAPLIST	Lists all joined or disjoined lines in a model (for models imported from CAD files).
GAPMERGE	Merges adjacent disjoined lines (for models imported from CAD files).
GAPOPT	Sets preferences for the CAD import repair commands.
GAPPLOT	Plots all joined or disjoined lines (for models imported from CAD files).
LNCOLLAPSE	Collapse a line segment to a keypoint (for models imported from CAD files).
LNDETACH	Detaches lines from neighboring geometric entity (for models imported from CAD files).
LNFILL	Creates a straight line between two keypoints (for models imported from CAD files).
LNMERGE	Merges two or more connected line segments (for models imported from CAD files).
LNSPLIT	Splits a line segment into two line segments (for models imported from CAD files).
SARPLOT	Displays areas smaller than a specified size (for models imported from CAD files).
SLPPLOT	Displays line loops smaller than a specified size (for models imported from CAD files).
SLSPLOT	Displays line segments smaller than a specified size (for models imported from CAD files).
VCVFILL	Fills cavities and bosses in volumes (for models imported from CAD files).

Table 2.56 Sections

These PREP7 commands manage sections.

PRSSOL	Prints BEAM188 and BEAM189 section results.
SDELETE	Deletes cross sections from the ANSYS database.
SECDATA	Describes the geometry of a beam section.
SECJOINT	Defines local coordinate systems at the nodes that form the element.
/SECLIB	Sets the default section library path for the **SECREAD** command.
SECLOCK	Specifies locks on the element degrees of freedom.
SECNUM	Sets the element section attribute pointer.

These PREP7 commands manage sections.

SECOFFSET	Defines the section offset for beam cross sections.
SECPLOT	Plots the geometry of a beam section to scale.
SECREAD	Reads a customized beam section library or a user-defined beam section mesh into ANSYS.
SECSTOP	Specifies stops on the element degrees of freedom.
SECTYPE	Associates section type information with a section ID number.
SECWRITE	Creates an ASCII file containing user mesh section information.
SLIST	Summarizes the section properties for all defined sections in the current session of ANSYS.

Table 2.57 Morphing

These PREP7 commands are used to adjust the finite element mesh in the non-structural regions to coincide with the deflections of the structural regions.

MORPH	Turns morphing on or off.
DAMORPH	Move nodes in selected areas to conform to structural displacements.
DEMORPH	Move nodes in selected elements to conform to structural displacements.
DVMORPH	Move nodes in selected volumes to conform to structural displacements.

Table 2.58 Trefftz Domain

These PREP7 commands are used in the Trefftz method for modeling the open domain.

TZAMESH	Meshes the areas of a volume to create Trefftz nodes.
TZDELE	Deletes the Trefftz superelement, associated constraint equations and all supporting Trefftz files.
TZEGEN	Generates a Trefftz domain substructure and defines a Trefftz superelement for use in electrostatic analysis.

Table 2.59 Perfectly Matched Layers

This PREP7 command is used to create perfectly matched layers (PMLs) designed to absorb high frequency waves.

PMLOPT	Defines perfectly matched layers (PMLs) for a high-frequency analysis.
PMLSIZE	Determines number of PML layers.

Table 2.60 Fluid-Solid Interaction

These PREP7 commands are used to setup a fluid-solid interaction analysis.

FSAN	Turns a fluid-solid interaction analysis on or off.
FSCO	Sets convergence values for a fluid-solid interaction analysis.
FSDT	Sets time step increment for a fluid-solid interaction analysis.
FSIN	Specifies the interface load transfer option for a fluid-solid interaction analysis.
FSIT	Sets the maximum number of stagger iterations for a fluid-solid interaction analysis.
FSOR	Specifies analysis order for a fluid-solid interaction analysis.
FSOU	Sets output frequency for a fluid-solid interaction analysis.

These PREP7 commands are used to setup a fluid-solid interaction analysis.

FSRE	Sets relaxation values for a fluid-solid interaction analysis.
FSSTAT	Lists the settings for a fluid-solid interaction analysis.
FSTI	Sets end time for a fluid-solid interaction analysis.
FSTR	Specifies static or transient analyses for a fluid-solid interaction analysis.

2.6. SOLUTION Commands

These commands are used to load and solve the model. The commands are grouped by functionality.

Table 2.61 Analysis Options

These SOLUTION commands are used to set general analysis options.

ABEXTRACT	Extracts the alpha-beta damping multipliers for Rayleigh damping.
ADAMS	Performs solutions and writes flexible body information to a modal neutral file.
ADAPT	Adaptively meshes and solves a model.
ANTYPE	Specifies the analysis type and restart status.
BCSOPTION	Sets memory option for the sparse solver.
CECHECK	Check constraint equations and couplings for rigid body motions.
CHECK	Checks current database items for completeness.
CMATRIX	Performs electrostatic field solutions and calculates the self and mutual capacitances between multiple conductors.
CMSOPT	Specifies component mode synthesis (CMS) analysis options.
CNCHECK	Lists the initial status of your contact pairs.
CUTCONTROL	Controls time-step cutback during a nonlinear solution.
CYCOPT	Specifies the harmonic index solution for a cyclic symmetry analysis.
DMPEXT	Extracts modal damping coefficients in a specified frequency range.
DSPROC	Specifies number of processors for a distributed solution.
DSOPT	Sets options for the Distributed Domain Server; this command is only valid for sites with a license for the Parallel Performance for ANSYS add-on.
EMATWRITE	Forces the writing of all the element matrices to File.EMAT.
EQSLV	Specifies the type of equation solver.
ERESX	Specifies extrapolation of integration point results.
ESCHECK	Perform element shape checking for a selected element set.
ESSOLV	Performs a coupled electrostatic-structural analysis.
EXPASS	Specifies an expansion pass of an analysis.
FSRS	Specifies time or load step number for restart of a fluid-structure interaction analysis.
FSSOLV	Performs a coupled steady-state fluid-structural analysis.
GAUGE	Gauges the problem domain for an edge-element formulation.
HFEIGOPT	Specifies high frequency electromagnetic modal analysis options.
HFPA	Specifies a radiation scan angle for a phased array antenna analysis.
HFPCSWP	Calculates the propagating constants of a transmission line or waveguide over a frequency range.
HFSCAT	Specifies a high-frequency scattering analysis.
HFSWEEP	Performs a harmonic response for a high-frequency electromagnetic wave guide analysis.
LMATRIX	Calculates the differential inductance matrix and the total flux linkage in each coil for an N-winding system.
LUMPM	Specifies a lumped mass matrix formulation.
MONITOR	Controls contents of three variable fields in nonlinear solution monitor file.

These SOLUTION commands are used to set general analysis options.

MSAVE	Sets the memory saving feature for the PCG and DPCG solvers.
OPNCONTROL	Sets decision parameter for automatically increasing the time step interval.
PRECISION	Specifies machine precision for solvers (currently valid only for PCG solvers).
PSOLVE	Directs the program to perform a partial solution.
RATE	Specifies whether the effect of creep strain rate will be used in the solution of a load step.
SEEXP	Specifies options for the substructure expansion pass.
SEGEN	Automatically generate superelements.
SEOPT	Specifies substructure analysis options.
SOLCONTROL	Specifies whether to use optimized nonlinear solution defaults and some enhanced internal solution algorithms.
/SOLU	Enters the solution processor.
SOLVE	Starts a solution.
SPSCAN	Performs a harmonic analysis of a cellar unit over a range of angles and extracts the S-parameter.
SPSWP	Computes S-parameters over a frequency range and writes them to a file.
STAOPT	Specifies static analysis options.
TOFFST	Specifies the temperature offset from absolute zero to zero.

Table 2.62 p-Method Options

These SOLUTION commands are used to define options for p-Method analyses.

PCONV	Sets convergence values for p-method solutions.
PEXCLUDE	Specifies elements to be excluded from p-level escalations.
PINCLUDE	Specifies elements to be included in p-level escalations.
/PMETH	Activates the p-method solution options in the Graphical User Interface (GUI).
PMOPTS	Defines percentage tolerance for a p-Method solution.
PPRANGE	Specifies a range of p-level values for use in a p-method solution.

Table 2.63 Nonlinear Options

These SOLUTION commands are used to define options for nonlinear analyses.

ARCLEN	Activates the arc-length method.
ARCTRM	Controls termination of the arc-length solution.
BUCOPT	Specifies buckling analysis options.
CNVTOL	Sets convergence values for nonlinear analyses.
CRPLIM	Specifies the creep criterion for automatic time stepping.
/GST	Turns Graphical Solution Tracking (GST) on or off.
LNSRCH	Activates a line search to be used with Newton-Raphson.
MXPAND	Specifies the number of modes to expand and write for a modal or buckling analysis.
NCNV	Sets the key to terminate an analysis.
NEQIT	Specifies the maximum number of equilibrium iterations for nonlinear analyses.
NLDIAG	Sets nonlinear diagnostics functionality.

These SOLUTION commands are used to define options for nonlinear analyses.

NLDPOST	Gets element component information from diagnostic files.
NLGEOM	Includes large deformation effects in a static or full transient analysis.
NLHIST	Specify result items to track during solution.
NROPT	Specifies the Newton-Raphson options in a static or full transient analysis.
PRED	Activates a predictor in a nonlinear analysis.
PSTRES	Specifies whether prestress effects are calculated or included.
SSTIF	Activates stress stiffness effects in a nonlinear analysis.
SUBOPT	Specifies options for subspace iteration eigenvalue extraction.

Table 2.64 Dynamic Options

These SOLUTION commands are used to define options for dynamic analyses.

ALPHAD	Defines the mass matrix multiplier for damping.
BETAD	Defines the stiffness matrix multiplier for damping.
DMPRAT	Sets a constant damping ratio.
HARFRQ	Defines the frequency range in the harmonic response analysis.
HREXP	Specifies the phase angle for the harmonic analysis expansion pass.
HROPT	Specifies harmonic analysis options.
HROUT	Specifies the harmonic analysis output options.
LVSCALE	Scales the load vector for mode superposition analyses.
MDAMP	Defines the damping ratios as a function of mode.
MDPLOT	Plots frequency-dependent modal damping coefficients.
MODOPT	Specifies modal analysis options.
MXPAND	Specifies the number of modes to expand and write for a modal or buckling analysis.
RIGID	Specifies known rigid body modes (if any) of the model.
SUBOPT	Specifies options for subspace iteration eigenvalue extraction.
TIMINT	Turns on transient effects.
TINTP	Defines transient integration parameters.
TRNOPT	Specifies transient analysis options.

Table 2.65 Spectrum Options

These SOLUTION commands are used to define options for spectrum analyses.

ADDAM	Specifies the acceleration spectrum computation constants for the analysis of shock resistance of shipboard structures.
COVAL	Defines PSD cospectral values.
CQC	Specifies the complete quadratic mode combination method.
DSUM	Specifies the double sum mode combination method.
FREQ	Defines the frequency points for the SV vs. FREQ tables.
GRP	Specifies the grouping mode combination method.
NRLSUM	Specifies the Naval Research Laboratory (NRL) sum mode combination method.
PFACT	Calculates participation factors for the PSD or multi-point response spectrum table.

These SOLUTION commands are used to define options for spectrum analyses.

PSDCOM	Specifies the power spectral density mode combination method.
PSDFRQ	Defines the frequency points for the input spectrum vs. FREQ tables of PSD and multi-point spectrum analyses.
PSDGRAPH	Displays input PSD curves
PSDRES	Controls solution output written to the results file from a PSD analysis.
PSDSPL	Defines a partially correlated excitation in a PSD analysis.
PSDUNIT	Defines the type of PSD or multi-point response spectrum.
PSDVAL	Defines PSD or multi-point response spectrum values.
PSDWAV	Defines a wave propagation excitation in a PSD analysis.
QDVAL	Defines PSD quadspectral values.
ROCK	Specifies a rocking response spectrum.
SED	Defines the excitation direction for a single-point response spectrum.
SPOPT	Selects the spectrum type and other spectrum options.
SRSS	Specifies the square root of sum of squares mode combination method.
SV	Defines spectrum values to be associated with frequency points.
SVTYP	Defines the type of single-point response spectrum.
VDDAM	Specifies the velocity spectrum computation constants for the analysis of shock resistance of shipboard structures.

Table 2.66 Load Step Options

These SOLUTION commands are used to define options for individual load steps.

AUTOTS	Specifies whether to use automatic time stepping or load stepping.
CECMOD	Modifies the constant term of a constraint equation during solution.
DELTIM	Specifies the time step sizes to be used for this load step.
EXPSOL	Specifies the solution to be expanded for reduced analyses.
HMAGSOLV	Specifies 2-D or axisymmetric harmonic magnetic solution options and initiates the solution.
KBC	Specifies stepped or ramped loading within a load step.
KUSE	Specifies whether or not to reuse the triangularized matrix.
MAGOPT	Specifies options for a 3-D magnetostatic field analysis.
MAGSOLV	Specifies magnetic solution options and initiates the solution.
MODE	Specifies the harmonic loading term for this load step.
NSUBST	Specifies the number of substeps to be taken this load step.
NUMEXP	Specifies solutions to be expanded from reduced analyses.
TIME	Sets the time for a load step.
TREF	Defines the reference temperature for the thermal strain calculations.
TSRES	Defines an array of keytimes at which the time-stepping strategy changes.
UPCOORD	Modifies the coordinates of the active set of nodes, based on the current displacements.
USRCAL	Allows user-solution subroutines to be activated or deactivated.
WRFULL	Stops solution after assembling global matrices.

Table 2.67 Solid Constraints

These SOLUTION commands are used to define constraints on the solid model.

DA	Defines symmetry or antisymmetry DOF constraints on areas.
DADELE	Deletes DOF constraints on an area.
DALIST	Lists the DOF constraints on an area.
DK	Defines DOF constraints at keypoints.
DKDELE	Deletes DOF constraints at a keypoint.
DKLIST	Lists the DOF constraints at keypoints.
DL	Defines symmetry or antisymmetry DOF constraints on lines.
DLDELE	Deletes DOF constraints on a line.
DLLIST	Lists DOF constraints on a line.
DTRAN	Transfers solid model DOF constraints to the finite element model.

Table 2.68 Solid Forces

These SOLUTION commands are used to define forces on the solid model.

FK	Defines force loads at keypoints.
FKDELE	Deletes force loads at a keypoint.
FKLIST	Lists the forces at keypoints.
FTRAN	Transfers solid model forces to the finite element model.

Table 2.69 Solid Surface Loads

These SOLUTION commands are used to define surface loads on the solid model.

SFA	Specifies surface loads on the selected areas.
SFADELE	Deletes surface loads from areas.
SFALIST	Lists the surface loads for the specified area.
SFL	Specifies surface loads on lines of an area.
SFLDELE	Deletes surface loads from lines.
SFLLIST	Lists the surface loads for lines.
SFTRAN	Transfer the solid model surface loads to the finite element model.

Table 2.70 Solid Body Loads

These SOLUTION commands are used to define body loads on the solid model.

BFA	Defines a body force load on an area.
BFADELE	Deletes body force loads on an area.
BFALIST	Lists the body force loads on an area.
BFK	Defines a body force load at a keypoint.
BFKDELE	Deletes body force loads at a keypoint.
BFKLIST	Lists the body force loads at keypoints.
BFL	Defines a body force load on a line.
BFLDELE	Deletes body force loads on a line.
BFLLIST	Lists the body force loads on a line.

These SOLUTION commands are used to define body loads on the solid model.

BFTRAN	Transfers solid model body force loads to the finite element model.
BFV	Defines a body force load on a volume.
BFVDELE	Deletes body force loads on a volume.
BFVLIST	Lists the body force loads on a volume.

Table 2.71 Inertia

These SOLUTION commands are used to define inertial loads on the model.

ACEL	Specifies the linear acceleration of the structure.
CGLOC	Specifies the origin location of the acceleration coordinate system.
CGOMGA	Specifies the rotational velocity of the global origin.
CMDOMEGA	Specifies the rotational acceleration of an element component about a user-defined rotational axis.
CMOMEGA	Specifies the rotational velocity of an element component about a user-defined rotational axis.
DCGOMG	Specifies the rotational acceleration of the global origin.
DOMEGA	Specifies the rotational acceleration of the structure.
IRLF	Specifies that inertia relief calculations are to be performed.
OMEGA	Specifies the rotational velocity of the structure.

Table 2.72 Miscellaneous Loads

These SOLUTION commands are for miscellaneous load definition and control.

BIOT	Calculates the Biot-Savart source magnetic field intensity.
FMAGBC	Applies force and torque boundary conditions to an element component.
HFPORT	Specifies input data for waveguide or transmission line ports.
IC	Specifies initial conditions at nodes.
ICDELE	Deletes initial conditions at nodes.
ICE	Specifies initial conditions on elements.
ICEDELE	Deletes initial conditions on elements.
ICELIST	Lists initial conditions on elements.
ICLIST	Lists the initial conditions.
ISFILE	Reads an initial stress state from a file into ANSYS.
MPCHG	Changes the material number attribute of an element.
OUTPR	Controls the solution printout.
OUTRES	Controls the solution data written to the database.
PGRAPH	Specifies the location from which graphics data will be retrieved for viewing.
PGSAVE	Creates a PowerGraphics (PGR) file from results data.
PGWRITE	Writes selected solution data to the PGR file for faster post processing access.
PLWAVE	Specifies a free-space time-harmonic incident plane electromagnetic wave in the global Cartesian coordinate system.
RESCONTROL	Controls file writing for multiframe restarts.
SBCLIST	Lists solid model boundary conditions.
SBCTRAN	Transfers solid model loads and boundary conditions to the FE model.

These SOLUTION commands are for miscellaneous load definition and control.

WSPRINGS	Creates weak springs on corner nodes of a bounding box of the currently selected elements.

Table 2.73 Load Step Operations

These SOLUTION commands are used to write and solve multiple load steps.

LSCLEAR	Clears loads and load step options from the database.
LSDELE	Deletes load step files.
LSREAD	Reads load and load step option data into the database.
LSSOLVE	Reads and solves multiple load steps.
LSWRITE	Writes load and load step option data to a file.

Table 2.74 Master DOF

These SOLUTION commands are used to define master degrees of freedom.

M	Defines master degrees of freedom for reduced and superelement generation analyses.
MDELE	Deletes master degrees of freedom.
MGEN	Generates additional MDOF from a previously defined set.
MLIST	Lists the MDOF of freedom.
TOTAL	Specifies automatic MDOF generation.

Table 2.75 Gap Conditions

These SOLUTION commands are used to define gaps for transient dynamic analyses.

GP	Defines a gap condition for transient analyses.
GPDELE	Deletes gap conditions.
GPLIST	Lists the gap conditions.

Table 2.76 Birth and Death

These SOLUTION commands are used for the birth and death option for elements.

EALIVE	Reactivates an element (for the birth and death capability).
EKILL	Deactivates an element (for the birth and death capability).
ESTIF	Specifies the matrix multiplier for deactivated elements.

Table 2.77 FE Constraints

These SOLUTION commands are used to define constraints on the finite element model.

D	Defines DOF constraints at nodes.
DCUM	Specifies that DOF constraint values are to be accumulated.
DDELE	Deletes degree of freedom constraints.
DJ	Specify displacement (or rotation) boundary conditions on the components of relative motion of a joint element.
DJDELE	Deletes displacement (or rotation) boundary conditions on the components of relative motion of a joint element.

These SOLUTION commands are used to define constraints on the finite element model.

DJLIST	Lists boundary conditions applied to joint elements.
DLIST	Lists DOF constraints.
DSCALE	Scales DOF constraint values.
DSYM	Specifies symmetry or antisymmetry DOF constraints on nodes.
GSBDATA	Specifies the constraints or applies the load at the ending point for the generalized plane strain element option.
GSLIST	When using generalized plane strain, lists the input data or solutions.
LDREAD	Reads results from the results file and applies them as loads.

Table 2.78 FE Forces

These SOLUTION commands are used to define nodal loads on the finite element model.

F	Specifies force loads at nodes.
FCUM	Specifies that force loads are to be accumulated.
FDELE	Deletes force loads on nodes.
FJ	Specify forces or moments on the components of the relative motion of a joint element.
FJDELE	Deletes forces (or moments) on the components of the relative motion of a joint element.
FJLIST	Lists forces and moments applied on joint elements.
FLIST	Lists force loads on the nodes.
FSCALE	Scales force load values in the database.

Table 2.79 FE Surface Loads

These SOLUTION commands are used to define surface loads on the finite element model.

SF	Specifies surface loads on nodes.
SFBEAM	Specifies surface loads on beam elements.
SFCUM	Specifies that surface loads are to be accumulated.
SFDELE	Deletes surface loads.
SFE	Specifies surface loads on elements.
SFEDELE	Deletes surface loads from elements.
SFELIST	Lists the surface loads for elements.
SFFUN	Specifies a varying surface load.
SFGRAD	Specifies a gradient (slope) for surface loads.
SFLIST	Lists surface loads.
SFSCALE	Scales surface loads on elements.

Table 2.80 FE Body Loads

These SOLUTION commands are used to define body loads on the finite element model.

BF	Defines a nodal body force load.
BFCUM	Specifies that nodal body force loads are to be accumulated.
BFDELE	Deletes nodal body force loads.
BFE	Defines an element body force load.

These SOLUTION commands are used to define body loads on the finite element model.

BFECUM	Specifies whether to ignore subsequent element body force loads.
BFEDELE	Deletes element body force loads.
BFELIST	Lists the element body force loads.
BFESCAL	Scales element body force loads.
BFLIST	Lists the body force loads on nodes.
BFSCALE	Scales body force loads at nodes.
BFUNIF	Assigns a uniform body force load to all nodes.
LDREAD	Reads results from the results file and applies them as loads.
RIMPORT	Imports initial stresses from an explicit run into ANSYS.
TUNIF	Assigns a uniform temperature to all nodes.

Table 2.81 Status

These SOLUTION commands are for use with the STAT command.

ATYPE	Specifies "Analysis types" as the subsequent status topic.
BIOOPT	Specifies "Biot-Savart options" as the subsequent status topic.
DEACT	Specifies "Element birth and death" as the subsequent status topic.
DYNOPT	Specifies "Dynamic analysis options" as the subsequent status topic.
GAP	Specifies "Reduced transient gap conditions" as the subsequent status topic.
GENOPT	Specifies "General options" as the subsequent status topic.
INRTIA	Specifies "Inertial loads" as the subsequent status topic.
LSOPER	Specifies "Load step operations" as the subsequent status topic.
MASTER	Specifies "Master DOF" as the subsequent status topic.
NLOPT	Specifies "Nonlinear analysis options" as the subsequent status topic.
OUTOPT	Specifies "Output options" as the subsequent status topic.
SMBODY	Specifies "Body loads on the solid model" as the subsequent status topic.
SMCONS	Specifies "Constraints on the solid model" as the subsequent status topic.
SMFOR	Specifies "Forces on the solid model" as the subsequent status topic.
SMSURF	Specifies "Surface loads on the solid model" as the subsequent status topic.
SOLUOPT	Specifies "Solution options" as the subsequent status topic.
SPTOPT	Specifies "Spectrum analysis options" as the subsequent status topic.

Table 2.82 Explicit Dynamics

These SOLUTION commands are used for an explicit dynamic analysis.

EDADAPT	Activates adaptive meshing in an explicit dynamic analysis.
EDALE	Assigns mesh smoothing to explicit dynamic elements that use the ALE formulation.
EDBVIS	Specifies global bulk viscosity coefficients for an explicit dynamic analysis.
EDCADAPT	Specifies adaptive meshing controls for an explicit dynamic analysis.
EDCPU	Specifies CPU time limit for an explicit dynamic analysis.
EDCSC	Specifies whether to use subcycling in an explicit dynamic analysis.
EDCTS	Specifies mass scaling and scale factor of computed time step for an explicit dynamic analysis.

These SOLUTION commands are used for an explicit dynamic analysis.

EDDAMP	Defines mass weighted (Alpha) or stiffness weighted (Beta) damping for an explicit dynamics model.
EDDRELAX	Activates initialization to a prescribed geometry or dynamic relaxation for the explicit analysis.
EDDUMP	Specifies output frequency for the explicit dynamic restart file (d3dump).
EDENERGY	Specifies energy dissipation controls for an explicit dynamic analysis.
EDFPLOT	Allows plotting of explicit dynamics forces.
EDGCALE	Defines global ALE controls for an explicit dynamic analysis.
EDHGLS	Specifies the hourglass coefficient for an explicit dynamic analysis.
EDHIST	Specifies time-history output for an explicit dynamic analysis.
EDHTIME	Specifies the time-history output interval for an explicit dynamic analysis.
EDINT	Specifies number of integration points for explicit shell and beam output.
EDIS	Specifies stress initialization in an explicit dynamic full restart analysis.
EDLOAD	Specifies loads for an explicit dynamic analysis.
EDOPT	Specifies the type of output for an explicit dynamic analysis.
EDOUT	Specifies time-history output (ASCII format) for an explicit dynamic analysis.
EDPL	Plots a time dependent load curve in an explicit dynamic analysis.
EDPVEL	Applies initial velocities to parts or part assemblies in an explicit dynamic analysis.
EDRC	Specifies rigid/deformable switch controls in an explicit dynamic analysis.
EDRD	Switches a part from deformable to rigid or from rigid to deformable in an explicit dynamic analysis.
EDRI	Defines inertia properties for a new rigid body that is created when a deformable part is switched to rigid in an explicit dynamic analysis.
EDRST	Specifies the output interval for an explicit dynamic analysis.
EDRUN	Specifies serial or parallel processing for an explicit dynamic analysis.
EDSHELL	Specifies shell computation controls for an explicit dynamic analysis.
EDSOLV	Specifies "explicit dynamics solution" as the subsequent status topic.
EDSTART	Specifies status (new or restart) of an explicit dynamic analysis.
EDTERM	Specifies termination criteria for an explicit dynamic analysis.
EDTP	Plots explicit elements based on their time step size.
EDVEL	Applies initial velocities to nodes or node components in an explicit dynamic analysis.
EDWRITE	Writes explicit dynamics input to an LS-DYNA input file.
REXPORT	Exports displacements from an implicit run to ANSYS LS-DYNA.

Table 2.83 FLOTRAN Checkout

These SOLUTION commands are used for a FLOTRAN analysis.

FLOCHECK	Sets up and runs a zero-iteration FLOTRAN analysis.

Table 2.84 Controls

This SOLUTION command is used to specify controls for electrostatic p-Method analyses.

PEMOPTS	Defines percentage tolerance and error estimation method for electrostatic p-Method solution.

Table 2.85 Radiosity

These SOLUTION commands are used to calculate the radiation view factors and to specify the solution parameters for the Radiosity solver method.

HEMIOPT	Specifies options for Hemicube view factor calculation.
RADOPT	Specifies Gauss-Seidel Radiosity Solver options.
SPCNOD	Defines a space node for radiation using the Radiosity method.
SPCTEMP	Defines a free-space ambient temperature for radiation using the Radiosity method.
STEF	Specifies Stefan-Boltzmann radiation constant.
V2DOPT	Specifies 2-D/axisymmetric view factor calculation options.
VFCALC	Computes and stores Hemicube view factors.
VFOPT	Specifies options for view factor file.
QSOPT	Specifies quasi static radiation options.

Table 2.86 Fluid-Structure Interaction

These SOLUTION commands are used for a fluid-solid interaction analysis.

FSAN	Turns a fluid-solid interaction analysis on or off.
FSCO	Sets convergence values for a fluid-solid interaction analysis.
FSDT	Sets time step increment for a fluid-solid interaction analysis.
FSIN	Specifies the interface load transfer option for a fluid-solid interaction analysis.
FSIT	Sets maximum number of iterations for a fluid-solid interaction analysis.
FSOR	Specifies analysis order for a fluid-solid interaction analysis.
FSOU	Sets output frequency for a fluid-solid interaction analysis.
FSRE	Sets relaxation values for a fluid-solid interaction analysis.
FSRS	Specifies time or load step number for restart of a fluid-structure interaction analysis.
FSSTAT	Lists the settings for a fluid-solid interaction analysis.
FSTI	Sets end time and load time for a fluid-solid interaction analysis.
FSTR	Specifies static or transient analyses for a fluid-solid interaction analysis.

Table 2.87 Multi-Field Analysis

These SOLUTION commands are used for Multi-field analysis.

MFANALYSIS	Turns a multifield analysis on or off.
MFBUCKET	Turns a bucket search on or off.
MFCALC	Specifies a calculation frequency for a field in a multifield analysis.
MFCLEAR	Deletes multifield analysis settings.
MFCMMAND	Captures field solution options in a command file.

These SOLUTION commands are used for Multi-field analysis.

MFCONV	Sets convergence values for a multifield analysis.
MFDTIME	Sets time step increment for a multifield analysis.
MFELEM	Defines a field by grouping element types.
MFEXTER	Defines external fields for a multifield analysis.
MFFNAME	Specifies a file name for a field in a multifield analysis.
MFIMPORT	Imports a new field into a current multifield analysis.
MFINTER	Specifies the interface load transfer interpolation option for a multifield analysis.
MFITER	Sets the maximum number of stagger iterations for a multifield analysis.
MFLIST	Lists the settings for a multifield analysis.
MFMAP	Calculates, saves, resumes, or deletes mapping data in a multifield analysis.
MFORDER	Specifies field solution order for a multifield analysis.
MFOUTPUT	Specifies results file output frequency for a multifield analysis.
MFRELAX	Sets relaxation values for a multifield analysis.
MFRSTART	Specifies a restart time for a multifield analysis.
MFSURFACE	Defines a surface load transfer for a multifield analysis.
MFTIME	Sets end time for a multifield analysis.
MFTOL	Turns normal distance checking on for surface mapping in a multifield analysis.
MFVOLUME	Defines volume load transfer across interface for a multi-field analysis.

2.7. POST1 Commands

These commands are used to postprocess the results with the database processor. The commands are grouped by functionality.

Table 2.88 Setup

These POST1 commands are used to put data into the database for postprocessing.

APPEND	Reads data from the results file and appends it to the database.
DETAB	Modifies element table results in the database.
DNSOL	Defines or modifies solution results at a node.
FILE	Specifies the data file where results are to be found.
HRCPLX	Computes and stores in the database the time-harmonic solution at a prescribed phase angle.
PGRSET	Defines the data set to be read from the PGR file.
/POST1	Enters the database results postprocessor.
RESET	Resets all POST1 or POST26 specifications to initial defaults.
SET	Defines the data set to be read from the results file.
SUBSET	Reads results for the selected portions of the model.

Table 2.89 Controls

These POST1 commands are used to control the calculations done for other commands.

AVPRIN	Specifies how principal and vector sums are to be calculated.
AVRES	Specifies how results data will be averaged when PowerGraphics is enabled.
/EFACET	Specifies the number of facets per element edge for PowerGraphics displays.
ERNORM	Controls error estimation calculations.
FORCE	Selects the element nodal force type for output.
INRES	Identifies the data to be retrieved from the results file.
LAYER	Specifies the element layer for which data are to be processed.
RSYS	Activates a coordinate system for printout or display of results.
SHELL	Selects a shell element or shell layer location for results output.

Table 2.90 Results

These POST1 commands are used to process results, such as DOF results, nodal stresses, and element summable and nonsummable miscellaneous data.

NSORT	Sorts nodal data.
NUSORT	Restores original order for nodal data.
PLCONV	Plots the convergence curve for specified items from a p-method solution.
PLDISP	Displays the displaced structure.
PLESOL	Displays the solution results as discontinuous element contours.
PLNSOL	Displays results as continuous contours.
PLSP	Displays S-parameters on an XY graph.
PLVECT	Displays results as vectors.
PPLOT	Displays an element plot indicating each element's final p-level.

These POST1 commands are used to process results, such as DOF results, nodal stresses, and element summable and nonsummable miscellaneous data.

PRCONV	Lists convergence values versus characteristic p-level.
PRESOL	Prints the solution results for elements.
PRJSOL	Print joint element output.
PRNLD	Prints the summed element nodal loads.
PRNSOL	Prints the nodal solution results.
PRRFOR	Used with the **FORCE** command. Prints the constrained node reaction solution.
PRRSOL	Prints the constrained node reaction solution.
PRSSOL	Prints BEAM188 and BEAM189 section results
PRVECT	Prints results as vector magnitude and direction cosines.
SUMTYPE	Sets the type of summation to be used in the following load case operations.

Table 2.91 Element Table

These POST1 commands are used to operate with the element table, which in turn is used to process results stored for each element, such as average stresses, heat fluxes, etc.

DESOL	Defines or modifies solution results at a node of an element.
ESORT	Sorts the element table.
ETABLE	Fills a table of element values for further processing.
EUSORT	Restores original order of the element table.
PLETAB	Displays element table items.
PLLS	Displays element table items as contoured areas along elements.
PLVECT	Displays results as vectors.
PRETAB	Prints the element table items.
PRVECT	Prints results as vector magnitude and direction cosines.
SABS	Specifies absolute values for element table operations.
SADD	Forms an element table item by adding two existing items.
SALLOW	Defines the allowable stress table for safety factor calculations.
SEXP	Forms an element table item by exponentiating and multiplying.
SFACT	Allows safety factor or margin of safety calculations to be made.
SFCALC	Calculates the safety factor or margin of safety.
SMAX	Forms an element table item from the maximum of two other items.
SMIN	Forms an element table item from the minimum of two other items.
SMULT	Forms an element table item by multiplying two other items.
SSUM	Calculates and prints the sum of element table items.
TALLOW	Defines the temperature table for safety factor calculations.
VCROSS	Forms element table items from the cross product of two vectors.
VDOT	Forms an element table item from the dot product of two vectors.

Table 2.92 Listing

These POST1 commands are used to control printed listings of results.

/FORMAT	Specifies format controls for tables.
/HEADER	Sets page and table heading print controls.

These POST1 commands are used to control printed listings of results.

IRLIST	Prints inertia relief summary table.
/PAGE	Defines the printout and screen page size.
PRERR	Prints SEPC and TEPC.
PRITER	Prints solution summary data.

Table 2.93 Animation

These POST1 commands are used to animate results.

ANCNTR	Produces an animated sequence of a contoured deformed shape.
ANCUT	Produces an animated sequence of Q-slices.
ANDATA	Produces a sequential contour animation over a range of results data.
ANDSCL	Produces an animated sequence of a deformed shape.
ANCYC	Applies a traveling wave animation to graphics data in a modal cyclic symmetry analysis.
ANDYNA	Produces an animated sequence of contour values through substeps.
/ANFILE	Saves or resumes an animation sequence to or from a file.
ANFLOW	Produces an animated sequence of particle flow in a flowing fluid or a charged particle traveling in an electric or magnetic field.
ANHARM	Produces a time-transient animated sequence of time-harmonic results (**ANTYPE**,HARMIC).
ANIM	Displays graphics data in animated form.
ANISOS	Produces an animated sequence of an isosurface.
ANMODE	Produces an animated sequence of a mode shape.
ANMRES	Performs animation of results over multiple results files in an explicit dynamic structural analysis or fluid flow analysis with remeshing.
ANTIME	Produces a sequential contour animation over a range of time.
TRTIME	Defines the options used for the **PLTRAC** (particle flow or charged particle trace) command.

Table 2.94 Path Operations

These POST1 commands are used for path operations.

PADELE	Deletes a defined path.
PAGET	Writes current path information into an array variable.
PAPUT	Retrieves path information from an array variable.
PARESU	Restores previously saved paths from a file.
PASAVE	Saves selected paths to an external file.
PATH	Defines a path name and establishes parameters for the path.
PCALC	Forms additional labeled path items by operating on existing path items.
PCROSS	Calculates the cross product of two path vectors along the current path.
PDEF	Interpolates an item onto a path.
PDOT	Calculates the dot product of two path vectors along the current path.
PLPAGM	Displays path items along the path geometry.
PLPATH	Displays path items on a graph.

These POST1 commands are used for path operations.

PLSECT	Displays membrane and membrane-plus-bending linearized stresses.
PMAP	Creates mapping of the path geometry by defining path interpolation division points.
PPATH	Defines a path by picking or defining nodes, or locations on the currently active working plane, or by entering specific coordinate locations.
PRANGE	Determines the path range.
PRPATH	Prints path items along a geometry path.
PRSECT	Calculates and prints linearized stresses along a section path.
PSEL	Selects a path or paths.
PVECT	Interpolates a set of items onto a path.

Table 2.95 Surface Operations

These POST1 commands are used to define an arbitrary surface and to develop results information for that surface.

SUCALC	Create new result data by operating on two existing result datasets on a given surface.
SUCR	Create a surface.
SUDEL	Delete geometry information as well as any mapped results for specified surface or for all selected surfaces.
SUEVAL	Perform operations on a mapped item and store result in a scalar parameter.
SUGET	Create and dimension an NPT row array parameter named PARM, where NPT is the number of geometry points in SurfName.
SUMAP	Map results onto selected surface(s).
SUPL	Plot specified SetName result data on all selected surfaces or on the specified surface.
SUPR	Print surface information.
SURESU	Resume surface definitions from a specified file.
SUSAVE	Save surface definitions and result items to a file.
SUSEL	Selects a subset of surfaces
SUVECT	Operate between two mapped result vectors.

Table 2.96 Load Case Calculations

These POST1 commands are used for combining results from different load steps.

LCABS	Specifies absolute values for load case operations.
LCASE	Reads a load case into the database.
LCDEF	Creates a load case from a set of results on a results file.
LCFACT	Defines scale factors for load case operations.
LCFILE	Creates a load case from an existing load case file.
LCOPER	Performs load case operations.
LCSEL	Selects a subset of load cases.
LCSUM	Specifies whether to process nonsummable items in load case operations.
LCWRITE	Creates a load case by writing results to a load case file.
LCZERO	Zeroes the results portion of the database.

These POST1 commands are used for combining results from different load steps.

RAPPND	Appends results data from the database to the results file.

Table 2.97 Magnetics Calculations

These POST1 commands are used for special purpose magnetics postprocessing.

CURR2D	Calculates current flow in a 2-D conductor.
EMAGERR	Calculates the relative error in an electrostatic or electromagnetic field analysis.
EMF	Calculates the electromotive force (emf), or voltage drop along a predefined path.
FLUXV	Calculates the flux passing through a closed contour.
FMAGSUM	Summarizes electromagnetic force calculations on element components.
FOR2D	Calculates magnetic forces on a body.
IMPD	Calculates the impedance of a conductor at a reference plane.
MMF	Calculates the magnetomotive force along a path.
PLF2D	Generates a contour line plot of equipotentials.
POWERH	Calculates the rms power loss in a conductor or lossy dielectric.
QFACT	Calculates the quality factor for high-frequency electromagnetic resonators.
REFLCOEF	Calculates the voltage reflection coefficient (REFLC), standing wave ratio (VSWR), and return loss (RL) in a COAX fed device; at postprocessing of an HF electromagnetic analysis.
SENERGY	Determines the stored magnetic energy or co-energy.
SPARM	Calculates scattering (S) parameters between ports of a waveguide.
TORQ2D	Calculates torque on a body in a magnetic field.
TORQC2D	Calculates torque on a body in a magnetic field based on a circular path.
TORQSUM	Summarizes electromagnetic torque calculations on element components.

Table 2.98 Fatigue

These POST1 commands are used for fatigue analyses.

FE	Defines a set of fatigue event parameters.
FELIST	Lists the fatigue event parameters.
FL	Defines a set of fatigue location parameters.
FLLIST	Lists the fatigue location parameters.
FP	Defines the fatigue S vs. N and Sm vs. T tables.
FPLIST	Lists the property table stored for fatigue evaluation.
FS	Stores fatigue stress components at a node.
FSDELE	Deletes a stress condition for a fatigue location, event, and loading.
FSLIST	Lists the stresses stored for fatigue evaluation.
FSNODE	Calculates and stores the stress components at a node for fatigue.
FSPLOT	Displays a fatigue stress item for a fatigue location and event.
FSSECT	Calculates and stores total linearized stress components.
FTCALC	Performs fatigue calculations for a particular node location.
FTSIZE	Defines the fatigue data storage array.
FTWRITE	Writes all currently stored fatigue data on a file.

Table 2.99 Trace Points

These POST1 commands are used to trace particle motions in a flow stream.

PLTRAC	Displays a particle flow or charged particle trace on an element display.
TRPDEL	Deletes particle flow or charged particle trace points.
TRPLIS	Lists the particle flow or charged particle trace points.
TRPOIN	Defines a point through which a particle flow or charged particle trace will travel.

Table 2.100 FLOTRAN Processing

These POST1 commands are used to postprocess the FLOTRAN CFD results.

FLREAD	Reads the residual file written by the FLOTRAN CFD option.

Table 2.101 Special Purpose

These POST1 commands are used for various special purposes.

BFINT	Activates the body force interpolation operation.
CBDOF	Activates cut boundary interpolation (for submodeling).
CMSFILE	Specifies the component mode synthesis (CMS) results files to include when plotting the mode shape of an assembly.
/CYCEXPAND	Graphically expands displacements, stresses and strains of a cyclically symmetric model partially or though the full 360 degrees by combining the *real* (original nodes and elements) and *imaginary* (duplicate nodes and elements) parts of the solution.
CYCPHASE	Provides tools for determining minimum and maximum possible result values from frequency couplets produced in a modal cyclic symmetry analysis.
/EXPAND	Allows the creation of a larger graphic display than that which is represented by the actual finite element analysis model.
EXPAND	Displays the results of a modal cyclic symmetry analysis.
FSUM	Sums the nodal force and moment contributions of elements.
HFANG	Defines or displays spatial angles of a spherical radiation surface for antenna parameter calculations.
HFARRAY	Defines phased array antenna characteristics.
HFNEAR	Calculates the electromagnetic field at points exterior to the visual equivalent source surface (Maxwell surface flag).
HFSYM	Sets symmetry planes for the computation of high-frequency electromagnetic fields in the near and far field domains (beyond the finite element region).
INTSRF	Integrates nodal results on an exterior surface.
KCALC	Calculates stress intensity factors in fracture mechanics analyses.
NFORCE	Sums the nodal forces and moments of elements attached to nodes.
PLCRACK	Displays cracking and crushing locations in SOLID65 elements.
PLHFFAR	Displays the far electromagnetic field, radar cross section, or antenna pattern results.
PRHFFAR	Prints the far electromagnetic field, radar cross section, or antenna pattern results.
RMFLVEC	Writes eigenvectors of fluid nodes to a file for use in damping parameter extraction.

These POST1 commands are used for various special purposes.

SPOINT	Defines a point for moment summations.

Table 2.102 Status

These POST1 commands are for use with the STAT command.

CALC	Specifies "Calculation settings" as the subsequent status topic.
DATADEF	Specifies "Directly defined data status" as the subsequent status topic.
DEFINE	Specifies "Data definition settings" as the subsequent status topic.
DISPLAY	Specifies "Display settings" as the subsequent status topic.
LCCALC	Specifies "Load case settings" as the subsequent status topic.
POINT	Specifies "Point flow tracing settings" as the subsequent status topic.
PRINT	Specifies "Print settings" as the subsequent status topic.
SORT	Specifies "Sort settings" as the subsequent status topic.
SPEC	Specifies "Miscellaneous specifications" as the subsequent status topic.

Table 2.103 Failure Criteria

These POST1 commands are for use with the failure criteria.

FC	Provides failure criteria information and activates a data table to input temperature-dependent stress and strain limits.
FCCHECK	Checks both the strain and stress input criteria for all materials.
FCDELE	Deletes previously defined failure criterion data for the given material.
FCLIST	To list what the failure criteria is that you have input.

2.8. POST26 Commands

These commands are used to postprocess the results with the time-history processor. The commands are grouped by functionality.

Table 2.104 Set Up

These POST26 commands are used to store data for processing.

ANSOL	Specifies averaged nodal data to be stored from the results file in the solution coordinate system.
DATA	Reads data records from a file into a variable.
EDREAD	Reads explicit dynamics output into variables for time-history postprocessing.
ESOL	Specifies element data to be stored from the results file.
FILE	Specifies the data file where results are to be found.
GAPF	Defines the gap force data to be stored in a variable.
GSSOL	Specifies which results to store from the results file when using generalized plane strain.
JSOL	Stores the displacement, rotation, reaction forces, and moments for the joint element.
NSOL	Specifies nodal data to be stored from the results file.
NSTORE	Defines which time points are to be stored.
NUMVAR	Specifies the number of variables allowed in POST26.
/POST26	Enters the time-history results postprocessor.
RESET	Resets all POST1 or POST26 specifications to initial defaults.
RFORCE	Specifies the total reaction force data to be stored.
/RGB	Specifies the RGB color values for indices and contours.
SOLU	Specifies solution summary data per substep to be stored.
STORE	Stores data in the database for the defined variables.
TIMERANGE	Specifies the time range for which data are to be stored.
VARDEL	Deletes a variable (GUI).
VARNAM	Names (or renames) a variable.

Table 2.105 Controls

These POST26 commands are used to control the calculations of other commands.

CFACT	Defines complex scaling factors to be used with operations.
FORCE	Selects the element nodal force type for output.
LAYERP26	Specifies the element layer for which data are to be stored.
SHELL	Selects a shell element or shell layer location for results output.
TVAR	Changes time to the cumulative iteration number.

Table 2.106 Operations

These POST26 commands are used to perform operations on the stored variables.

ABS	Forms the absolute value of a variable.
ADD	Adds variables.

These POST26 commands are used to perform operations on the stored variables.

ATAN	Forms the arctangent of a complex variable.
CLOG	Forms the common log of a variable
CONJUG	Forms the complex conjugate of a variable.
DERIV	Differentiates a variable.
EXP	Forms the exponential of a variable.
FILLDATA	Fills a variable by a ramp function.
IMAGIN	Forms an imaginary variable from a complex variable.
INT1	Integrates a variable.
LARGE	Finds the largest (the envelope) of three variables.
NLOG	Forms the natural log of a variable.
PROD	Multiplies variables.
QUOT	Divides two variables.
REALVAR	Forms a variable using only the real part of a complex variable.
SMALL	Finds the smallest of three variables.
SQRT	Forms the square root of a variable.

Table 2.107 Display

These POST26 commands are used to display the results.

PLCPLX	Specifies the part of a complex variable to display.
PLTIME	Defines the time range for which data are to be displayed.
PLVAR	Displays up to ten variables in the form of a graph.
SPREAD	Turns on a dashed tolerance curve for the subsequent curve plots.
XVAR	Specifies the X variable to be displayed.

Table 2.108 Listing

These POST26 commands are used to produce tabular listings of the results.

EXTREM	Lists the extreme values for variables.
LINES	Specifies the length of a printed page.
NPRINT	Defines which time points stored are to be listed.
PRCPLX	Defines the output form for complex variables.
PRTIME	Defines the time range for which data are to be listed.
PRVAR	Lists variables vs. time (or frequency).

Table 2.109 Special Purpose

These POST26 commands are used for various special purposes.

CVAR	Computes covariance between two quantities.
PMGTRAN	Summarizes electromagnetic results from a transient analysis.
RESP	Generates a response spectrum.
RPSD	Computes response power spectral density (PSD).
SMOOTH	Allows smoothing of noisy data and provides a graphical representation of the data.

These POST26 commands are used for various special purposes.

VGET	Moves a variable into an array parameter vector.
VPUT	Moves an array parameter vector into a variable.

Table 2.110 Status

These POST26 commands are for use with the STAT command.

DEFINE	Specifies "Data definition settings" as the subsequent status topic.
OPERATE	Specifies "Operation data" as the subsequent status topic.
PLOTTING	Specifies "Plotting settings" as the subsequent status topic.
PRINT	Specifies "Print settings" as the subsequent status topic.

2.9. AUX2 Commands

These commands are used to examine the contents of binary files produced by the program. The commands are grouped by functionality.

Table 2.111 Binary Files

These AUX2 commands are used to dump the contents of binary files.	
/AUX2	Enters the binary file dumping processor.
DUMP	Dumps the contents of a binary file.
FILEAUX2	Specifies the binary file to be dumped.
FORM	Specifies the format of the file dump.
HBMAT	Writes an assembled global matrix in Harwell-Boeing format.

2.10. AUX3 Commands

The auxiliary processor **/AUX3** allows you to operate on results files by deleting sets or by changing values.

Table 2.112 Results Files

These commands are related to the /AUX3 command.	
/AUX3	Enters the results file editing processor.
COMPRESS	Deletes all specified sets.
DELETE	Specifies sets in the results file to be deleted before postprocessing.
FILEAUX3	Specifies the results file to be edited.
LIST	Lists out the sets in the results file.
MODIFY	Changes the listed values of the data in a set.
UNDELETE	Removes results sets from the group of sets selected for editing.

2.11. AUX12 Commands

These commands are used to create a radiation substructure for use in thermal analyses. The commands are grouped by functionality.

Table 2.113 Radiation Substructures

These AUX12 commands are used to add radiation effects to a thermal analysis.	
/AUX12	Enters the radiation matrix generation processor.
EMIS	Specifies the emissivity as a material property.
GEOM	Defines the geometry specifications for the radiation matrix calculation.
MPRINT	Specifies that radiation matrices are to be printed.
SPACE	Defines a space node for radiation.
STEF	Specifies Stefan-Boltzmann radiation constant.
VFQUERY	Queries and prints element Hemicube view factors and average view factor.
VTYPE	Specifies the viewing procedure used to determine the form factors.
WRITE	Writes the radiation matrix file.

2.12. AUX15 Commands

These commands are used to read in an IGES file for analysis in ANSYS. The commands are grouped by functionality.

Table 2.114 IGES

These AUX15 commands read an IGES format file into PREP7 data.	
/AUX15	Enters the IGES file transfer processor.
IGESIN	Transfers IGES data from a file into ANSYS.
IOPTN	Controls options relating to importing a model.

2.13. RUNSTATS Commands

These commands are used to estimate run times and file sizes. The commands are grouped by functionality.

Table 2.115 Run Statistics Estimator

These RUNSTATS commands are used to calculate estimates of ANSYS program run times and file sizes.	
RALL	Calculates solver statistics and run time estimates.
RFILSZ	Estimates file sizes.
RITER	Supplies an estimate of the number of iterations for time estimates.
RMEMRY	Prints memory statistics for the current model.
RSPEED	Supplies system performance information for use in time estimates.
RSTAT	Prints the FE model statistics of the model.
RTIMST	Prints runtime estimates.
/RUNST	Enters the run statistics processor.
RWFRNT	Generates wavefront statistics and memory requirements.

2.14. OPTIMIZATION Commands

These commands are used for design optimization analyses. The commands are grouped by functionality.

Table 2.116 Specifications

These OPTIMIZATION commands set up the design optimization variables.

OPEQN	Controls curve fitting for the subproblem approximation method.
OPFACT	Defines the type of factorial evaluation to be performed.
OPFRST	Defines specifications for the first order optimization method.
OPGRAD	Specifies which design set will be used for gradient evaluation.
OPKEEP	Specifies whether to save the best-set results and database file.
OPLOOP	Specifies controls for optimization looping.
OPPRNT	Activates detailed optimization summary printout.
OPRAND	Defines the number of iterations for a random optimization.
OPSUBP	Defines number of iterations for subproblem approximation method.
OPSWEEP	Specifies the reference point and number of evaluation points for a sweep generation.
/OPT	Enters the design optimizer.
OPTYPE	Specifies the optimization method to be used.
OPUSER	Defines specifications for user-supplied external optimization.
OPVAR	Specifies the parameters to be treated as optimization variables.
TOCOMP	Defines single or multiple compliance as the topological optimization function.
TODEF	Defines parameters for and initializes topological optimization.
TOFREQ	Defines single or mean frequency formulation as the topological optimization function.
TOTYPE	Specifies solution method for topological optimization.
TOVAR	Specifies the objective and constraints for the topological optimization problem.

Table 2.117 Operations

These OPTIMIZATION commands operate on the design optimization variables.

OPADD	Forms a set of optimization parameters by adding two sets.
OPCLR	Clears the optimization database.
OPDEL	Deletes optimization design sets.
OPMAKE	Creates a design set using active scalar parameter values.
OPSEL	Selects design sets for subsequent optimization looping.

Table 2.118 Files

These OPTIMIZATION commands operate on the design optimization files.

OPANL	Defines the analysis file to be used for optimization looping.
OPDATA	Identifies the file where optimization data is to be saved.
OPRESU	Reads optimization data into the optimization database.
OPSAVE	Writes all optimization data to a file.

Table 2.119 Run

These OPTIMIZATION commands perform the design optimization analysis.

OPEXE	Initiates optimization looping.
TOEXE	Executes one topological optimization iteration.
TOLOOP	Execute several topological optimizations iterations.

Table 2.120 Display

These OPTIMIZATION commands are used to display the optimization results as plots.

OPLFA	Displays the results of a factorial evaluation.
OPLGR	Graphs the results of a gradient evaluation.
OPLIST	Displays the parameters for design sets.
OPLSW	Graphs the results of a global sweep generation.
OPRFA	Prints the results of a factorial evaluation.
OPRGR	Prints the results of a gradient evaluation.
OPRSW	Prints the results of a global sweep generation.
PLVAROPT	Displays up to ten parameters in the form of a graph.
PRVAROPT	Lists up to ten optimization parameters.
TOGRAPH	Plots iteration solution of topological optimization.
TOLIST	Lists all topological optimization functions currently defined.
TOPLOT	Plot current topological density distribution.
TOPRINT	Print iteration solution history of topological optimization.
TOSTAT	Displays topological optimization status and results information.
XVAROPT	Specifies the parameter to be used as the X-axis variable.

2.15. VARIATIONAL TECHNOLOGY Commands

These commands are used for Variational Technology products.

Table 2.121 DesignXplorer VT

These Variational Technology commands are part of the ANSYS DesignXplorer VT.

SXCLR	Specifies the part of the Variational Technology database to be cleared.
SXDISC	Defines an element component as a discrete input variable for the DesignXplorer VT.
SXFREQ	Defines the frequency as input variable for the FS Module or the DesignXplorer VT.
SXMETH	Defines the solution methods options for the DesignXplorer VT.
SXMP	Defines a material property as an input variable for the DesignXplorer VT.
SXPOST	Launches the DesignXplorer VT postprocessing application.
SXREAL	Defines a real constant property as an input variable for the DesignXplorer VT.
SXRFIL	Specifies the file to which DesignXplorer VT results are written.
SXRSLT	Defines a result quantity for the DesignXplorer VT.
SXSEC	Defines a section property as an input variable for the FS Module or the DesignXplorer VT.

These Variational Technology commands are part of the ANSYS DesignXplorer VT.

SXSTAT	Print the status of the DesignXplorer VT definitions and settings into a separate window.
SXVMOD	Modifies the status or current value of an input variable for the DesignXplorer VT.

Table 2.122 FS Module

These Variational Technology commands are part of the ANSYS Frequency Sweep VT.

SXFREQ	Defines the frequency as input variable for the FS Module or the DesignXplorer VT.

2.16. PROBABILISTIC Design Commands

These commands are used for probabilistic design analyses. The commands are grouped by functionality.

Table 2.123 Deterministic Model

The following PDS commands allow the specification of the deterministic model.

PDANL	Defines the analysis file to be used for probabilistic looping.

Table 2.124 Probabilistic Preprocessing

The following PDS commands allow the specification and visualization of the probabilistic model.

PDCORR	Specifies the correlation between two random input variables.
PDINQR	Retrieves a value and stores it as a user-parameter.
PDPLOT	Plots the distribution curves of a defined random input variable.
PDVAR	Specifies the parameters to be treated as probabilistic design variables.

Table 2.125 Probabilistic Methods

The following PDS commands allow the specification of the probabilistic methods and their options used in a probabilistic analysis.

PDDOEL	Defines design of experiment levels for an individual random input variable.
PDDMCS	Specifies options for Monte Carlo Simulations.
PDMETH	Specifies the probabilistic analysis method.
PDUSER	Specifies options for user-specified sampling methods.

Table 2.126 Run Probabilistic Analysis

The following PDS commands allow the execution of a probabilistic analysis.

PDEXE	Executes the probabilistic analysis.

Table 2.127 Probabilistic Postprocessing

The following PDS commands allow the probabilistic postprocessing and the visualization of the probabilistic results

PDCDF	Plots the cumulative distribution function.
PDCMAT	Prints the correlation coefficient matrix.

The following PDS commands allow the probabilistic postprocessing and the visualization of the probabilistic results

PDHIST	Plots the frequency histogram.
PDPINV	Prints the result of the inversion of a probability.
PDPROB	Prints a probability result.
PDROPT	Specifies the options for an HTML report.
PDSCAT	Plots a scatter graph.
PDSENS	Plots the probabilistic sensitivities.
PDSHIS	Plots sample history values.
PDWRITE	Generates an HTML report for the probabilistic analysis.

Table 2.128 Probabilistic Database

The following PDS commands provide access to the probabilistic database.

PDCLR	Clears the probabilistic design database.
PDRESU	Reads the probabilistic model data and stores it in the database.
PDSAVE	Writes the probabilistic model data to a file.

Table 2.129 Response Surface

The following PDS commands allow the evaluation, visualization, and use of the response surfaces.

RSFIT	Fit a response surface for an output parameter in a solution set.
RSPLOT	Plot a response surface.
RSPRNT	Print a response surface.
RSSIMS	Performs Monte Carlo simulations on response surface(s).

Table 2.130 Auxiliary Commands and Information

The following commands are helpful for using the PDS.

/PDS	Enters the probabilistic design system.

2.17. DISPLAY Program Commands

These commands are used for the DISPLAY program. The DISPLAY program is a companion program to ANSYS, used for recovering graphics displays produced within ANSYS. The commands are grouped by functionality.

Table 2.131 Set Up

These DISPLAY commands are used to set up the DISPLAY program.

/CMAP	Changes an existing or creates a new color mapping table.
/DEVDISP	Controls graphics device options.
FILEDISP	Specifies the file containing the graphics data.
HELPDISP	Displays help information on DISPLAY program commands.
NOCOLOR	Removes color from graphics displays.
/SEG	Allows graphics data to be stored in the local terminal memory.
/SHOWDISP	Defines the display driver name.
TRANS	Reformats File.GRPH for improved performance with plotters.

Table 2.132 Driver Options

These DISPLAY commands are used to specify the graphics driver and options for subsequent plots.

TERM	Specifies various terminal driver options.

Table 2.133 Action

These DISPLAY commands are used to produce the plots and exit the program.

FINISH	Exits normally from a processor.
PLOT	Forms a display.
STAT	Displays the status of database settings.

2.18. REDUCED Order Modeling Commands

These commands are used for reduced order modeling analyses. The commands are grouped by functionality.

Table 2.134 Set Up

These commands are used to save or resume the ROM database.

RMRESUME	Resumes ROM data from a file.
RMSAVE	Saves ROM data to file.

Table 2.135 Preparation

These commands are used to create the input files for the ROM Generation Pass.

RMNDISP	Extracts neutral plane displacements from a test load or element load solution for the ROM method.
RMNEVEC	Extracts neutral plane eigenvectors from a modal analysis for the ROM method.

Table 2.136 Generation Pass

These commands are used to create a reduced order model.

RMALIST	Lists all defined master nodes for a ROM method.
RMANL	Assigns model database, dimensionality, and operating direction for the ROM method.
RMASTER	Defines master nodes for the ROM method.
RMCAP	Defines lumped capacitance pairs between conductors C1 and C2 for a ROM method.
RMCLIST	Lists all lumped capacitances defined.
RMMLIST	Lists all mode specifications for the ROM method.
RMMRANGE	Defines and edits various modal parameters for the ROM method.
RMMSELECT	Selects modes for the ROM method.
RMPORDER	Defines polynomial orders for ROM functions.
RMRGENERATE	Performs fitting procedure for all ROM functions to generate response surfaces.
RMROPTIONS	Defines options for ROM response surface fitting.
RMRPLOT	Plots response surface of ROM function or its derivatives with respect to the dominant mode(s).
RMRSTATUS	Prints status of response surface for ROM function.

These commands are used to create a reduced order model.

RMSMPLE	Runs finite element solutions and obtains sample points for the ROM method.
RMXPORT	Exports ROM model to external VHDL-AMS simulator.

Table 2.137 Use Pass

These commands use the reduced order model in an analysis.

DCVSWP	Performs a DC voltage sweep on a ROM element.
RMLVSCALE	Defines element load vector scaling for a ROM use pass.
RMUSE	Activates ROM use pass for ROM elements.

Chapter 3: Command Dictionary

This chapter contains a dictionary of the ANSYS commands, listed in alphabetical order. The star (*) and slash (/) of the star and slash commands are ignored for alphabetization (for example, the **/SHOW** command appears between the **SHELL** and **SHPP** commands). As in a dictionary, keywords are located at the top of each page (in the printed version only) indicating the first and last commands contained on that page.

The following section documents the components of a command description.

3.1. Components of a Command Description

SAMPLECMD, *Key*

Specifies whether to use automatic time stepping or load stepping.

SOLUTION: Load Step Options

MP ME ST DY <> PR EM <> FL PP ED

Argument Descriptions

Key
Automatic time stepping key:

OFF --
Do not use automatic time stepping

ON --
Use automatic time stepping.

Default: No automatic time stepping.

Notes

Specifies whether to use automatic time stepping (or load stepping) over this load step. If *Key* = ON, both time step prediction and time step bisection will be used. Used only if *DTIME* (specified on the **DELTIM** command) is less than the time span or conversely, if *NSBSTP* (on the **NSUBST** command) is greater than one.

This command is also valid in PREP7.

Product Restrictions

In ANSYS Professional, *Key* is automatically set to ON and cannot be changed.

Menu Paths

Main Menu> Preprocessor >Loads> Load Step Opts> Time/Frequenc> Freq and Substps
Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time - Time Step
Main Menu> Solution> Load Step Opts> Time/Frequenc> Freq and Substps
Main Menu> Solution> Load Step Opts> Time/Frequenc> Time - Time Step

3.1.1. Features Documented in ANSYS Commands

The first line of the command description shows the command name followed by the argument names (if any). The second line summarizes the command function. The summary is *not* intended to be all inclusive:

SAMPLECMD, *Key*

Specifies whether to use automatic time stepping or load stepping.

Listed on the next line are codes that will help you find other commands in the program with related functionality.

SOLUTION: Load Step Options

MP ME ST DY <> PR EM <> FL PP ED

"SOLUTION: Load Step Options" is the code in the example above. There is at least one group code for each command. The first portion of the code before the colon (:) tells which of the tables in Chapter 2, “Command Groupings” the command can be found in. These are major groupings, such as PREP7 or APDL commands. In the example above, SOLUTION commands are found in Section 2.6: SOLUTION Commands. Most of these major groupings are processors, and in those cases it is implied that the command can be entered only when in that processor. Any exceptions are noted under "Notes" later in the description.

The second portion of the code, after the colon, is the subtable in Chapter 2, “Command Groupings” containing the command. The subtables list commands that relate to each other in function. In the example, the code "Load Step Options" means that other commands related to load step options can be found in that subtable of Section 2.6: SOLUTION Commands (Table 2.66: “Load Step Options”).

If you are viewing this manual in the ANSYS Help System, just click on the code to follow the link to the corresponding table. Then click on any of the commands in the table, to follow the link to its description.

If you are reading the printed version, simply turn to the appropriate table in Chapter 2, “Command Groupings”. The table lists the page number where the command is documented.

The next line displays a series of product codes, which may contain all of the ANSYS product codes:

MP ME ST DY <> PR EM <> FL PP ED

or a subset of them:

MP ME ST <> <> PR EM <> <> PP ED

For more information on product codes, see Section 1.2.1.1: Product Codes.

Following this "product code" line is the description of all arguments, if any, of the command:

Key
: Automatic time stepping key:

  OFF --
  : Do not use automatic time stepping (default).

  ON --
  : Use automatic time stepping.

The argument is described and, where necessary, valid choices for the argument are described. Many arguments list defaults, which are the values assumed for *that* argument if you *enter the command but leave the argument blank*. On the other hand, shown after the argument descriptions is often another default:

Command Default

No automatic time stepping.

This is the *command default*. This is the specification assumed by the program if you *do not enter the command at all*. Only commands that set specifications (*specification* commands) have defaults listed. Commands that cause some action, such as performing some calculation, are called *action* commands and simply do not perform the action if the command is not entered. Defaults are not listed for action commands.

Following the default listing are any notes about the command. These notes expand on the summary description given up near the command format, and describe any other behavior, restrictions, suggestions, etc. of that command:

Notes

Specifies whether to use automatic time stepping (or load stepping) over this load step. If *Key* = ON, both time step prediction and time step bisection will be used. Used only if *DTIME* (specified on the **DELTIM** command) is less than the time span or conversely, if *NSBSTP* (on the **NSUBST** command) is greater than one.

This command is also valid in PREP7.

If the command behaves differently, or has restrictions, in any of the various ANSYS products, those differences are described in the *Product Restrictions* section:

In ANSYS Professional, *Key* is automatically set to ON and cannot be changed.

Product Restrictions

In ANSYS Professional, *Key* is automatically set to ON and cannot be changed.

Menu Paths

Main Menu >Preprocessor> Loads> Load Step Opts> Time/Frequenc> Freq and Substps
Main Menu >Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time - Time Step
Main Menu> Solution> Load Step Opts> Time/Frequenc >Freq and Substps
Main Menu> Solution> Load Step Opts> Time/Frequenc> Time - Time Step

These menu paths are intended as guides to help you find the commands in the GUI. Be aware, however, that often the paths are valid only if some other command has been previously issued. For example, you won't be able to follow a path to transient analysis options if you have not selected a transient analysis in the first place.

Not all commands are directly accessible through the menu (although some of them may be generated indirectly by some menu function). Such cases are indicated by the following statement in the menu path listing:

Menu Paths

This command cannot be accessed directly in the menu.

Connection Commands

~CAT5IN, *Name*, *Extension*, *Path*, *Entity*, *FMT*, *NOCL*, *NOAN*
Transfers a .CATPart file into the ANSYS program.

CAD Import

MP ME ST DY <> PR EM <> FL PP ED

Name
The name of a valid **.CATPart** file, created with CATIA Version 5.0. The first character of the file name must be an alphanumeric.

Extension
The extension for the file. The default extension is **.CATPart**.

Path
The path name of the directory in which the file resides enclosed in single quotes. The default path name is the current working directory.

Entity
Entity to be imported.

SOLIDS
Solids only, imported as ANSYS volumes (default).

SURFACES
Surfaces only, imported as ANSYS areas.

WIREFRAME
Wireframe only, imported as ANSYS lines.

ALL
All entities. Use this option when the file contains different types of entities.

FMT
The format in which ANSYS will store the model.

0
Neutral format (default). Defeaturing after import is restricted.

1
Solid format; this allows defeaturing after import.

NOCL
Remove tiny objects.

0
Remove tiny objects without checking model validity (default).

1
Do not remove tiny objects.

NOAN
Perform an analysis of the model.

0
Analyze the model (default).

1
Do not analyze the model.

Notes

More information on importing CATIA Version 5 parts is available in Section 2.2: ANSYS Connection for CATIA V5 in the *ANSYS Connection User's Guide*.

Menu Paths

File> Import> CATIA5

~CATIAIN, *Name*, *Extension*, *Path*, - -, - -, *BLANK*, - -

Transfers a CATIA model into the ANSYS program.

CAD Import

MP ME ST DY <> PR EM <> FL PP ED

Name
The name of a valid CATIA model, created with CATIA 4.x or lower. The first character of the file name must be an alphanumeric.

Extension
The extension for the file. The default extension is **.model**.

Path
The path name of the directory in which the file resides, enclosed in single quotes. The default path name is the current working directory.

- -
Unused field.

- -
Unused field.

BLANK
Sets whether to import "blanked" entities.

0
Does not import "blanked" (suppressed) CATIA entities (default).

1
Imports "blanked" entities. The portions of CATIA data that were suppressed will be included in the import.

- -
Unused field.

Notes

More information on importing CATIA parts is available in Section 2.1: ANSYS Connection for CATIA V4 in the *ANSYS Connection User's Guide*.

Menu Paths

File> Import> CATIA

~CIFIN, *Filename*, *Ext*, *Dir*, *Scale*, *Offset*

Transfers an electronic layer geometry file in CIF format into the ANSYS program.

CAD Import

MP ME ST DY <> PR EM <> FL PP ED

Filename
The name of valid CIF file.

Ext
Extension for the CIF file.

Dir
Directory.

Scale
Scale input factor, defaults to 1.

Offset
Layer offset, defaults to 1000.

Notes

Each layer has area, line, and keypoint components named A_Layer, L_Layer, and K_Layer where "Layer" is the layer name on the CIF file. An assembly named "Layer" is created from all layer components. The material attribute number MAT (**MAT** command) is set to the layer number in the CIF file.

More information on importing CIF parts is available in Section 2.3: ANSYS Connection for CIF in the *ANSYS Connection User's Guide*.

Menu Paths

File> Import> CIF

~HEAL, *Name*, *Extension*, *Path*, *Method*, *Check*

"Heal" a CAD file, then read it into ANSYS.

CAD Import

MP ME ST DY <> PR EM <> FL PP ED

Name
The file name.

Extension
The file extension, such as **x_t**, **iges**, **prt**, etc. Valid extensions are determined by licensing.

Path
The full path to the directory in which the file resides, enclosed in single quotes. The working directory is the default value.

Method
The method used to convert the original file to ANSYS neutral file format. Valid arguments are ANSYS and CADFIX (the default). From the GUI, the method is chosen automatically from the license you use to start Connections.

Check
Sets validation method. Valid arguments are MESH (the default), BOOL (boolean), and NONE (no validation check).

Notes

The output is a **.anf** file.

Menu Paths

File> Import> Healing

~PARAIN, *Name*, *Extension*, *Path*, *Entity*, *FMT*, *Scale*
Transfers a Parasolid file into the ANSYS program.

CAD Import

MP ME ST DY <> PR EM <> FL PP ED

Name
The name of a valid Parasolid file. The first character of the file name must be an alphanumeric.

Extension
The extension for the file. The default extension is **.x_t** on a PC or **.xmt_txt** on a Unix system. Parasolid files are compatible across systems, and do not need to be renamed to be used on another platform.

Path
The path name of the directory in which the file resides, enclosed in single quotes. The default path name is the current working directory.

Entity
Entity to be imported:

SOLIDS
Solids only, imported as ANSYS volumes (default)

SURFACES
Surfaces only, imported as ANSYS areas.

WIREFRAME
Wireframe only, imported as ANSYS lines.

ALL
All entities. Use this option when the file contains more than one type of entity.

FMT
Sets the format in which ANSYS will store the model

0
Neutral format (default). Defeaturing after import is restricted. Use this option if you need to scale a model to a specific unit of measure (other than meters).

1
Solid format; this allows defeaturing after import.

Scale
Allows scaling for the model

0
Do not rescale the model; retain the default Parasolid setting of meters (default).

1
Scale the model if warranted by the model size.

Very small models will be scaled by the factor of 10 or 100 to increase the chance of successful import; the scaling factor used is displayed in the output window and in the **.para_log** file. Because scaling changes the dimensions of the model, you must apply loads and material properties appropriately. If the model cannot be properly scaled, the analysis may fail.

If you need to scale your model to a specific set of measurements, set FMT = 0, then use either the **VLSCALE**, **ARSCALE** or **LSSCALE** command to select a different unit of measure.

Notes

More information on importing Parasolid parts is available in Section 2.4: ANSYS Connection for Parasolid in the *ANSYS Connection User's Guide*.

Menu Paths

File> Import> PARA

~PROEIN, *Name*, *Extension*, *Path*, *Proecomm*, *FMT*

Transfers a Pro/ENGINEER part into the ANSYS program.

CAD Import

MP ME ST DY <> PR EM <> FL PP ED

Name
The name of the Pro/ENGINEER part to be imported, which cannot exceed 64 characters in length and must begin with an alphanumeric character. Special characters such as & - and * and spaces are not permitted in the part name.

Extension
The general Pro/ENGINEER extension format is **prt**. The assembly extension format is **asm**

Path
Full path name to the directory containing the part. The default is the current working directory. Neither the path name nor the file name should contain a space.

Proecomm
The name of the version of Pro/ENGINEER you are using. **pro22** is the default version number; earlier releases are also supported. Note that the full path name to the **pro** command should *not* be used here since the path name should have been defined in the **PATH** variable. The Pro/ENGINEER command name is set by the **ANSYS_PROE_CMD** environment variable.

FMT
The format in which ANSYS will store the model.

0
Neutral format (default). Defeaturing after import is restricted.

1
Solid format; this allows defeaturing after import.

Notes

More information on importing Pro/ENGINEER parts is available in Section 2.5: ANSYS Connection for Pro/ENGINEER in the *ANSYS Connection User's Guide*.

Menu Paths

File> Import> Pro/E

~SATIN, *Name*, *Extension*, *Path*, *Entity*, *FMT*, *NOCL*, *NOAN*

Transfers a .SAT file into the ANSYS program.

CAD Import

MP ME ST DY <> PR EM <> FL PP ED

Name
The name of a valid **.SAT** file, created with a supported version of ACIS. The first character of the file name must be an alphanumeric. See File Names in the *ANSYS Commands Reference* for more information about ANSYS file naming conventions.

Extension
The extension for the file. The default extension is **.sat**.

Path
The path name of the directory in which the file resides enclosed in single quotes. The default path name is the current working directory.

Entity
Entity to be imported.

SOLIDS
Solids only, imported as ANSYS volumes (default).

SURFACES
Surfaces only, imported as ANSYS areas.

WIREFRAME
Wireframe only, imported as ANSYS lines.

ALL
All entities. Use this option when the file contains different types of entities.

FMT
The format in which ANSYS will store the model.

0
Neutral format (default). Defeaturing after import is restricted.

1
Solid format; this allows defeaturing after import.

NOCL
Remove tiny objects.

0
Remove tiny objects without checking model validity (default).

1
Do not remove tiny objects.

NOAN
Perform an ACIS analysis of the model.

0
Analyze the model (default).

1
Do not analyze the model.

Note — *NOCL* and *NOAN* are not supported in the ANSYS GUI.

Notes

More information on importing SAT parts is available in Section 2.6: ANSYS Connection for SAT in the *ANSYS Connection User's Guide*.

Menu Paths

File> Import> SAT

~UGIN, *Name*, *Extension*, *Path*, *Entity*, *LAYER*, *FMT*

Transfers a Unigraphics part into the ANSYS program.

CAD Import

MP ME ST DY <> PR EM <> FL PP ED

Name
The file name of the Unigraphics part to be imported, which cannot exceed 64 characters in length. The path name must begin with an alphanumeric character. Special characters such as &, -, and * are not permitted in the part name.

Extension
The Unigraphics part file extension. The default is **.prt**.

Path
The full path name to the directory containing the part, enclosed in single quotes; for example, **'/ug_parts'**. The default is the current working directory.

Entity
Entity to be imported.

0 or Solid
Solids only, imported as ANSYS volumes (the default).

1 or Surface
Surfaces only, imported as ANSYS areas.

2 or Wireframe
Wireframe only, imported as ANSYS lines.

3 or All
All entities. Use this option when the part contains entities that may not be attached to each other, such as a solid in one location and a surface in another.

LAYER
The number(s) assigned to the layer(s) to be imported. You can import one layer or a range of layers (designated by hyphens). Defaults to 1-256 (all layers).

FMT
The format in which ANSYS will store the model.

0
Neutral format (default). Defeaturing after import is restricted.

1
Solid format; this allows defeaturing after import.

Notes

More information on importing Unigraphics parts is available in Section 2.7: ANSYS Connection for Unigraphics in the *ANSYS Connection User's Guide*.

Menu Paths

File> Import> UG

A Commands

A, *P1*, *P2*, *P3*, *P4*, *P5*, *P6*, *P7*, *P8*, *P9*, *P10*, *P11*, *P12*, *P13*, *P14*, *P15*, *P16*, *P17*, *P18*
Defines an area by connecting keypoints.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

P1, *P2*, *P3*, *P4*, *P5*, *P6*, *P7*, *P8*, *P9*, *P10*, *P11*, *P12*, *P13*, *P14*, *P15*, *P16*, *P17*, *P18*
List of keypoints defining the area (18 maximum if using keyboard entry). At least 3 keypoints must be entered. If *P1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

Notes

Keypoints (*P1* through *P18*) must be input in a clockwise or counterclockwise order around the area. This order also determines the positive normal direction of the area according to the right-hand rule. Existing lines between adjacent keypoints will be used; missing lines are generated "straight" in the active coordinate system and assigned the lowest available numbers [**NUMSTR**]. If more than one line exists between two keypoints, the shorter one will be chosen. If the area is to be defined with more than four keypoints, the required keypoints and lines must lie on a constant coordinate value in the active coordinate system (such as a plane or a cylinder). Areas may be redefined only if not yet attached to a volume. Solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>Through KPs

---

AADD, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*
Adds separate areas to create a single area.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*
Numbers of areas to be added. If *NA1* = ALL, add all selected areas and ignore *NA2* to *NA9*. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

Notes

The areas must be coplanar. The original areas (and their corresponding lines and keypoints) will be deleted by default. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. Concatenated entities are not valid with this command.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Add>Areas

AATT, *MAT*, *REAL*, *TYPE*, *ESYS*, *SECN*

Associates element attributes with the selected, unmeshed areas.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

MAT
The material number to be associated with selected, unmeshed areas.

REAL
The real constant set number to be associated with selected, unmeshed areas.

TYPE
The type number to be associated with selected, unmeshed areas.

ESYS
The coordinate system number to be associated with selected, unmeshed areas.

SECN
The section number to be associated with selected unmeshed areas.

Notes

Areas subsequently generated from the areas will also have these attributes. These element attributes will be used when the areas are meshed. If an area does not have attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT**, **REAL**, **TYPE**, **ESYS**, and **SECNUM** command settings. Reissue the **AATT** command (before areas are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association. If any of the arguments *MAT*, *REAL*, *TYPE*, *ESYS*, or *SECN* are defined as -1, then that value will be left unchanged in the selected set.

In some cases, ANSYS can proceed with an area meshing operation even when no logical element type has been assigned via **AATT**,,,*TYPE* or **TYPE**. For more information, see the discussion on setting element attributes in Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>All Areas
Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked Areas

ABEXTRACT, *MODE1*, *MODE2*

Extracts the alpha-beta damping multipliers for Rayleigh damping.

SOLUTION: Analysis Options

MP ME <> <> <> <> <> <> <> PP ED

MODE1
First mode number.

MODE2
Second mode number.

Notes

ABEXTRACT calls the command macro **DMPEXT** to extract the damping ratio of *MODE1* and *MODE2* and then computes the Alpha and Beta damping multipliers for use in a subsequent structural harmonic or transient analysis. Both numbers are stored in parameters ALPHADMP and BETADMP and can be applied using the **ALPHAD** and **BETAD** commands. Before calling **ABEXTRACT**, you must issue **RMFLVEC** to extract the modal displacements. In addition, a node component FLUN must exist from all FLUID136 nodes. See Chapter 16, " Thin Film Analysis" for more information on thin film analyses.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>ThinFilm>RayleighDamp

ABS, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*

Forms the absolute value of a variable.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA
Reference number of the variable to be operated on.

--, --
Unused fields.

Name
Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --
Unused fields.

FACTA
Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

Notes

The new variable is calculated as:

$$IR = | FACTA \times IA |$$

For a complex number (a + *i*b), the absolute value is the magnitude, where the *IA* values are obtained from:

$$\sqrt{a^2 + b^2}$$

See the *ANSYS, Inc. Theory Reference* for details.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Absolute Value

ACCAT, *NA1*, *NA2*

Concatenates multiple areas in preparation for mapped meshing.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*

Areas to be concatenated. If *NA1* = ALL, *NA2* will be ignored and all selected areas [**ASEL**] will be concatenated. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* is ignored).

Notes

Concatenates multiple, adjacent areas (the input areas) into one area (the output area) in preparation for mapped meshing. A volume that contains too many areas for mapped meshing can still be mapped meshed if some of the areas in that volume are first concatenated (see Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide* for details on mapped meshing restrictions).

Because of modeling restrictions that result from its use, **ACCAT** is meant to be used solely for meshing. Specifically, (a) the output area and any volumes that have the output area on their area list [**VLIST**] cannot be used as input to any other solid modeling operation (not even another **ACCAT** command); and (b) the output area cannot accept solid model boundary conditions [**DA**, **SFA**].

The output area (or volumes which contain it) will be meshed [**AMESH**, **VMESH**] by meshing the input areas, which themselves must be meshable. The output area from the **ACCAT** operation will be coincident with the input areas and the input areas will be retained. Consider the **AADD** command instead of **ACCAT** if you wish to delete the input areas. When an **ACCAT** command is issued, volume area lists [**VLIST**] that contain *all* of the input areas will be updated so that the volume area lists refer to the output area instead of the input area. Deletion of the output area [**ADELE**] effectively reverses the **ACCAT** operation and restores volume area lists to their original condition. **ACCAT** operations on pairs of adjacent four-sided areas automatically concatenate appropriate lines [**LCCAT**]; in all other situations, line concatenations must be addressed by the user.

You can use the **ASEL** command to select areas that were created by concatenation, and then follow it with an **ADELE**,ALL command to delete them. See Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide* for a discussion on how to easily select and delete concatenated areas in one step.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>Concatenate>Areas

---

ACEL, *ACELX*, *ACELY*, *ACELZ*

Specifies the linear acceleration of the structure.

SOLUTION: Inertia

MP ME ST <> <> PR <> <> FL PP ED

ACELX, *ACELY*, *ACELZ*

Linear acceleration of the structure in the global Cartesian X, Y, and Z axis directions.

Notes

Related commands for rotational effects are **CGLOC**, **CGOMGA**, **DCGOMG**, **DOMEGA**, and **OMEGA**.

Defines the linear acceleration of the structure in each of the global Cartesian axis directions. To simulate gravity (by using inertial effects), accelerate the structure in the direction opposite to gravity. For example, apply a positive *ACELY* to simulate gravity acting in the negative Y direction. Units are length/time^2.

Accelerations may be defined in analysis types **ANTYPE**,STATIC; **ANTYPE**,HARMIC (except reduced); **ANTYPE**,TRANS; and **ANTYPE**,SUBSTR. For all but the reduced transient dynamic (**ANTYPE**,TRANS) analysis, accelerations are combined with the element mass matrices to form a body force load vector term. The element mass matrix may be formed from a mass input constant or from a nonzero density (DENS) property, depending upon the element type. Units of acceleration and mass must be consistent to give a product of force units. For analysis type **ANTYPE**,HARMIC, the acceleration is assumed to be the real component with a zero imaginary component. For **ANTYPE**,TRANS (reduced), the acceleration is applied to the reduced mass matrix.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Gravity
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Gravity
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Gravity
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Gravity
Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Gravity
Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Gravity

ACLEAR, *NA1*, *NA2*, *NINC*

Deletes nodes and area elements associated with selected areas.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NINC*

Delete mesh for areas *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the mesh for all selected areas [**ASEL**] is deleted. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

Notes

Deletes *all* nodes and area elements associated with selected areas (regardless of whether the nodes or elements are selected). Nodes shared by adjacent meshed areas and nodes associated with non-area elements will not be deleted. Attributes assigned as a result of **AATT** are maintained. In the program's response to the command, if an area, line, or keypoint is tallied as "cleared," it means either its node or element reference was deleted.

Menu Paths

Main Menu>Preprocessor>Meshing>Clear>Areas

ADAMS, *NMODES*, *KSTRESS*, *KSHELL*

Performs solutions and writes flexible body information to a modal neutral file (Jobname.MNF) for use in an ADAMS analysis.

SOLUTION: Analysis Options

MP ME ST <> <> <> <> <> <> <> ED

NMODES
Number of normal modes to be written to **Jobname.MNF** file (no default).

KSTRESS
Specifies whether to write stress or strain results:

0
Do not write stress or strain results (default).

1
Write stress results.

2
Write strain results.

3
Write both stress and strain results.

KSHELL
Shell element output location. This option is valid only for shell elements.

0, 1
Shell top surface (default).

2
Shell middle surface.

3
Shell bottom surface.

Notes

ADAMS invokes a predefined ANSYS macro that solves a series of analyses and then writes the modal neutral file, **Jobname.MNF**. This file can be imported into the ADAMS program in order to perform a rigid body dynamics simulation. For detailed information on how to use the **ADAMS** command macro to create a modal neutral file, see Rigid Body Dynamics and the ANSYS-ADAMS Interface in the *ANSYS Advanced Analysis Techniques Guide*.

Before running the **ADAMS** command macro, you must specify the units with the **/UNITS** command. The interface points should be the only selected nodes when the command macro is initiated. (Interface points are nodes where constraints may be applied in ADAMS.) Only selected elements will be considered in the calculations.

By default, stress and strain data is transferred to the ADAMS program for all nodes, as specified by the *KSTRESS* value. If you want to transfer stress/strain data for only a subset of nodes, select the desired subset and create a node component named "STRESS" before running the **ADAMS** command macro. For example, you may want to select exterior nodes for the purpose of visualization in the ADAMS program.

The default filename for the modal neutral file is **Jobname.MNF**. In interactive (GUI) mode, you can specify a filename other than **Jobname.MNF**. In batch mode, there is no option to change the filename, and the modal neutral file is always written to **Jobname.MNF**.

Menu Paths

Main Menu>Solution>ADAMS Connection>Export to ADAMS

ADAPT, *NSOLN*, *STARGT*, *TTARGT*, *FACMN*, *FACMX*, *KYKPS*, *KYMAC*

Adaptively meshes and solves a model.

SOLUTION: Analysis Options

MP ME ST <> <> PR <> <> <> <> ED

NSOLN
Number of solutions allowed (1 or more) (defaults to 5).

STARGT
Target percentage for structural percent error in energy norm (SEPC) (defaults to 5). If -1, no target value is used.

TTARGT
Target percentage for thermal percent error in energy norm (TEPC) (defaults to 1). If -1, no target value is used.

FACMN
Minimum factor for the keypoint element size changes (defaults to 0.25).

FACMX
Maximum factor for the keypoint element size changes (defaults to 2.0).

KYKPS
Specifies whether element size is to be modified at selected keypoints:

0
Modify element size regardless of selected keypoint set (default).

1
Modify element size only at selected keypoints.

KYMAC
Specifies which user-written auxiliary macro files are to be used:

0
Ignore user-written auxiliary macro files, if any (default).

1
Use user-written auxiliary macro files (if they exist) as follows: Use **ADAPTMSH.MAC** instead of the default meshing command sequence. Use **ADAPTSOL.MAC** instead of the default solution command sequence (**/SOLU** ... **SOLVE** ... **FINISH**).

Notes

ADAPT invokes a predefined ANSYS macro for adaptive meshing and solution. The macro causes repeated runs of the PREP7, SOLUTION, and POST1 phases of the ANSYS program with mesh density refinements based upon the percentage error in energy norm. See the *ANSYS Advanced Analysis Techniques Guide* for additional details. After the adaptive meshing process is complete, the **ADAPT** macro automatically turns element shape checking on (**SHPP**,ON).

A copy of the macro, called **UADAPT.MAC**, is available on the ANSYS distribution medium (system dependent), and may be copied and modified by the user to suit a particular need. The modified file should be given a suitable name (*cmd*.MAC) and run as described above with the **ADAPT** command name replaced by your "*cmd*" name.

This command is also valid at the Begin level.

Menu Paths

Main Menu>Solution>Solve>Adaptive Mesh

ADD, *IR*, *IA*, *IB*, *IC*, *Name*, --, --, *FACTA*, *FACTB*, *FACTC*

Adds variables.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA, *IB*, *IC*
Reference numbers of the three variables to be operated on. If only two variables, leave *IC* blank. If only one, leave *IB* and *IC* blank.

Name
Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --
Unused fields.

FACTA, *FACTB*, *FACTC*
Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

Notes

Adds variables (up to three at once) according to the operation.

Menu Paths

Main Menu>Drop Test>Time History>Graph Variables
Main Menu>Drop Test>Time History>List Variables
Main Menu>TimeHist Postpro>Math Operations>Add

ADDAM, *AF*, *AA*, *AB*, *AC*, *AD*, *AMIN*

Specifies the acceleration spectrum computation constants for the analysis of shock resistance of shipboard structures.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

AF

Direction-dependent acceleration coefficient for elastic or elastic-plastic analysis option (default = 0).

AA, *AB*, *AC*, *AD*

Coefficients for the DDAM acceleration spectrum equations. See the *ANSYS, Inc. Theory Reference*. Default for these coefficients is zero.

AMIN

The minimum acceleration value in inch/sec^2. It defaults to 2316 inch/sec^2 which equals 6g, where *g* is acceleration due to gravity (*g* = 386 inch/sec^2).

Notes

This command specifies acceleration coefficients to analyze shock resistance of shipboard equipment. These coefficients are used to compute mode coefficients according to the equations given in the *ANSYS, Inc. Theory Reference*. The form of these equations is based on the Naval NRL Dynamic Design Analysis Method. This command, along with the **VDDAM** and **SED** commands, is used with the spectrum (**ANTYPE**,SPECTR) analysis as a special purpose alternative to the **SV**, **FREQ**, and **SVTYP** commands. The mass and length units of the model must be in pounds and inches, respectively.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>DDAM Options
Main Menu>Solution>Load Step Opts>Spectrum>DDAM Options

ADELE, *NA1*, *NA2*, *NINC*, *KSWP*

Deletes unmeshed areas.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NINC*

Delete areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and all selected areas [**ASEL**] are deleted. If *NA1* = P, graphical picking is enabled and all remaining

arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

KSWP

Specifies whether keypoints and lines are also to be deleted:

0

Delete areas only (default).

1

Delete areas, as well as keypoints and lines attached to specified areas but not shared by other areas.

Notes

An area attached to a volume cannot be deleted unless the volume is first deleted.

Menu Paths

Main Menu>Preprocessor>Modeling>Delete>Area and Below
Main Menu>Preprocessor>Modeling>Delete>Areas Only
Main Menu>Preprocessor>Modeling>Topo Repair>Delete>Area and Below
Main Menu>Preprocessor>Modeling>Topo Repair>Delete>Areas Only

---

ADGL, *NA1*, *NA2*, *NINC*

Lists keypoints of an area that lie on a parametric degeneracy.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NINC*

List keypoints that lie on a parametric degeneracy on areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL (default), *NA2* and *NINC* will be ignored and keypoints on all selected areas [**ASEL**] will be listed. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may be substituted in *NA1* (*NA2* and *NINC* will be ignored).

Notes

See the *ANSYS Modeling and Meshing Guide* for details on parametric degeneracies.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>List Degen Areas
Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>List Degen Areas

---

ADRAG, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NLP1*, *NLP2*, *NLP3*, *NLP4*, *NLP5*, *NLP6*

Generates areas by dragging a line pattern along a path.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*

List of lines in the pattern to be dragged (6 maximum if using keyboard entry). Lines should form a continuous pattern (no more than two lines connected to any one keypoint. If *NL1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). If *NL1* = ALL, all selected lines (except those that define the drag path) will be swept along the path. A component name may also be substituted for *NL1*.

NLP1, *NLP2*, *NLP3*, *NLP4*, *NLP5*, *NLP6*

List of lines defining the path along which the pattern is to be dragged (6 maximum if using keyboard entry). Must be a continuous set of lines.

Notes

Generates areas (and their corresponding keypoints and lines) by sweeping a given line pattern along a characteristic drag path. If the drag path consists of multiple lines, the drag direction is determined by the sequence in which the path lines are input (*NLP1*, *NLP2*, etc.). If the drag path is a single line (*NLP1*), the drag direction is from the keypoint on the drag line that is closest to the first keypoint of the given line pattern to the other end of the drag line.

The magnitude of the vector between the keypoints of the given pattern and the first path keypoint remains constant for all generated keypoint patterns and the path keypoints. The direction of the vector relative to the path slope also remains constant so that patterns may be swept around curves.

Keypoint, line, and area numbers are automatically assigned (beginning with the lowest available values [**NUMSTR**]). Adjacent lines use a common keypoint. Adjacent areas use a common line. For best results, the entities to be dragged should be orthogonal to the start of the drag path. Drag operations that produce an error message may create some of the desired entities prior to terminating.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Lines>Along Lines

---

AESIZE, *ANUM*, *SIZE*,

Specifies the element size to be meshed onto areas.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

ANUM

Area number of the area to which this element size specification applies. If *ANUM* = ALL, size applies to all selected areas. If *ANUM* = P, graphical picking is enabled. A component name may also be substituted for *ANUM*.

SIZE

Desired element size.

Notes

AESIZE allows control over the element sizing inside any area or on the face(s) of a volume.

SIZE controls element size on the interior of the area. For any line on the area not having its own size assignment and not controlled by keypoint size assignments, it specifies the element size along the line as well, so long as no adjacent area has a smaller size, which would take precedence. If the **AESIZE** governs the boundary and SmartSizing is on, the boundary size can be refined for curvature or proximity.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Areas>All Areas
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Areas>Clr Size
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Areas>Picked Areas

---

AFILLT, *NA1*, *NA2*, *RAD*

Generates a fillet at the intersection of two areas.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NA1
Number of the first intersecting area. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

NA2
Number of the second intersecting area.

RAD
Radius of fillet to be generated.

Notes

Generates an area of constant fillet radius at the intersection of two areas using a series of Boolean operations. Corresponding lines and keypoints are also generated. See **BOPTN** command for an explanation of the options available to Boolean operations. If areas do not initially intersect at a common line, use the **AINA** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Area Fillet

AFLIST

Lists the current data in the database.

PREP7: Database

MP ME ST DY <> PR EM <> FL PP ED

Notes

Lists the current data and specifications in the database. If batch, lists all appropriate data. If interactive, lists only summaries.

Menu Paths

Utility Menu>List>Other>Database Summary

AFSURF, *SAREA*, *TLINE*

Generates surface elements overlaid on the surface of existing solid elements and assigns the extra node as the closest fluid element node.

PREP7: Elements

MP ME <> <> <> PR <> <> <> <> ED

SAREA
Component name for the surface areas of the meshed solid volumes.

TLINE
Component name for the target lines meshed with fluid elements.

Notes

This command macro is used to generate surface effect elements overlaid on the surface of existing solid elements and, based on proximity, to determine and assign the extra node for each surface element. The underlying volumes of the solid region and the fluid lines must be meshed prior to calling this command macro.

The surface areas of the solid and the target lines of the fluid are grouped into components and named using the **CM** command. The names must be enclosed in single quotes (e.g., '*SAREA*') when the **AFSURF** command is manually typed in.

When using the GUI method, node and element components are created through the picking dialog boxes associated with this command.

The macro is applicable for the SURF152 and FLUID116 element types.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Area to Fluid
Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Line to Fluid
Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Node to Fluid

---

AGEN, *ITIME*, *NA1*, *NA2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

Generates additional areas from a pattern of areas.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

ITIME

Do this generation operation a total of *ITIME*s, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be more than 1 for generation to occur.

NA1, *NA2*, *NINC*

Generate areas from the pattern of areas *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is all selected areas [**ASEL**]. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

DX, *DY*, *DZ*

Keypoint location increments in the active coordinate system (--, D θ, DZ for cylindrical; --, D θ, -- for spherical).

KINC

Keypoint number increment between generated sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM

Specifies if elements and nodes are also to be generated:

0

Generate nodes and elements associated with the original areas, if they exist.

1

Do not generate nodes and elements.

IMOVE

Specifies whether to redefine the existing areas:

0

Generate new areas as requested with the *ITIME* argument.

1

Move original areas to new position, retaining the same keypoint numbers (*ITIME*, *KINC*, and *NOELEM* are ignored). If the original areas are needed in the original position (e.g., they may be attached to a volume), they are not moved, and new areas are generated instead. Meshed items corresponding to moved areas are also moved if not needed at their original position.

Notes

Generates additional areas (and their corresponding keypoints, lines and mesh) from a given area pattern. The MAT, TYPE, REAL, ESYS, and SECNUM attributes of the new areas are based upon the areas in the pattern and not upon the current settings of the pointers. End slopes of the generated lines remain the same (in the active coordinate system) as those of the given pattern. For example, radial slopes remain radial. Generations which produce areas of a size or shape different from the pattern (i.e., radial generations in cylindrical systems, radial and phi generations in spherical systems, and theta generations in elliptical systems) are not allowed. Solid modeling in a toroidal coordinate system is not recommended. Area and line numbers are automatically assigned, beginning with the lowest available values [**NUMSTR**].

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Areas
Main Menu>Preprocessor>Modeling>Move / Modify>Areas>Areas

AGLUE, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Generates new areas by "gluing" areas.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*
Numbers of the areas to be glued. If *NA1* = ALL, all selected areas will be glued (*NA2* to *NA9* will be ignored). If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

Notes

Use of the **AGLUE** command generates new areas by "gluing" input areas. The glue operation redefines the input areas so that they share lines along their common boundaries. The new areas encompass the same geometry as the original areas. This operation is only valid if the intersection of the input areas are lines along the boundaries of those areas. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to new entities generated.

The **AGLUE** command results in the merging of lines and keypoints at the common area boundaries. The lines and keypoints of the lower numbered area will be kept. This means one must be aware of area numbering when multiple **AGLUE** commands are applied to avoid any “ungluing” of geometry.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Glue>Areas

AINA, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Finds the intersection of areas.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*
Numbers of areas to be intersected. If *NA1* = ALL, *NA2* to *NA9* are ignored and the intersection of all selected areas is found. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

Notes

Finds the common (not pairwise) intersection of areas. The common intersection is defined as the regions shared (in common) by **all** areas listed on this command. New areas will be generated where the original areas intersect. If the regions of intersection are only lines, new lines will be generated instead. See the *ANSYS Modeling and*

Meshing Guide for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Common>Areas

---

AINP, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Finds the pairwise intersection of areas.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Numbers of areas to be intersected pairwise. If *NA1* = ALL, *NA2* to *NA9* are ignored and the pairwise intersection of all selected areas is found. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may be substituted for *NA1*.

Notes

Finds the pairwise intersection of areas. The pairwise intersection is defined as all regions shared by any two or more areas listed on this command. New areas will be generated where the original areas intersect pairwise. If the regions of pairwise intersection are only lines, new lines will be generated. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Pairwise>Areas

---

AINV, *NA*, *NV*

Finds the intersection of an area with a volume.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NA

Number of area to be intersected. If P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

NV

Number of volume to be intersected.

Notes

New areas will be generated where the areas intersect the volumes. If the regions of intersection are only lines, new lines will be generated instead. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN**

command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Area with Volume

---

AL, *L1*, *L2*, *L3*, *L4*, *L5*, *L6*, *L7*, *L8*, *L9*, *L10*

Generates an area bounded by previously defined lines.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

L1, *L2*, *L3*, *L4*, *L5*, *L6*, *L7*, *L8*, *L9*, *L10*
: List of lines defining area. The minimum number of lines is 3. The positive normal of the area is controlled by the direction of *L1* using the right-hand rule. A negative value of *L1* reverses the normal direction. If *L1* = ALL, use all selected lines with *L2* defining the normal (*L3* to *L10* are ignored and *L2* defaults to the lowest numbered selected line). If *L1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *L1*.

Notes

Lines may be input (once each) in any order and must form a simply connected closed curve. If the area is defined with more than four lines, the lines must also lie in the same plane or on a constant coordinate value in the active coordinate system (such as a plane or a cylinder).

Note — Solid modeling in a toroidal coordinate system is not recommended. Areas may be redefined only if not yet attached to a volume.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Lines

---

ALIST, *NA1*, *NA2*, *NINC*, *Lab*

Lists the defined areas.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NINC*
: List areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL (default), *NA2* and *NINC* are ignored and all selected areas [**ASEL**] are listed. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

Lab
: Determines what type of listing is used (one of the following):

(blank)
Prints information about all areas in the specified range.

HPT
Prints information about only those areas that contain hard points.

Notes

An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **AATT** command (and will not be reset to zero if the mesh is cleared); one listed as a negative value indicates that the attribute was assigned using the attribute pointer [**TYPE**, **MAT**, **REAL**, or **ESYS**] that was active during meshing (and will be reset to zero if the mesh is cleared). A "-1" in the "nodes" column indicates that the area has been meshed but there are no interior nodes. The area size is listed only if an **ASUM** command has been performed on the area.

Menu Paths

Utility Menu>List>Areas

ALLSEL, *LabT*, *Entity*

Selects all entities with a single command.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

LabT
Type of selection to be made:

ALL
Selects all items of the specified entity type and all items of lower entity types (default).

BELOW
Selects all items directly associated with and below the selected items of the specified entity type.

Entity
Entity type on which selection is based:

ALL
All entity types (default).

VOLU
Volumes.

AREA
Areas.

LINE
Lines.

KP
Keypoints.

ELEM
Elements.

NODE
Nodes.

Notes

ALLSEL is a convenience command that allows the user to select all items of a specified entity type or to select items associated with the selected items of a higher entity.

An entity hierarchy is used to decide what entities will be available in the selection process. This hierarchy from top to bottom is as follows: volumes, areas, lines, keypoints, elements, and nodes. The hierarchy may also be divided into two branches: the solid model and the finite element model. The label ALL selects items based on one branch only, while BELOW uses the entire entity hierarchy. For example, **ALLSEL**,ALL,VOLU selects all volumes, areas, lines, and keypoints in the data base. **ALLSEL**,BELOW,AREA selects all lines belonging to the selected areas; all keypoints belonging to those lines; all elements belonging to those areas, lines, and keypoints; and all nodes belonging to those elements.

The $ character should not be used after the **ALLSEL** command.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking
Utility Menu>Select>Everything
Utility Menu>Select>Everything Below>Selected Areas
Utility Menu>Select>Everything Below>Selected Elements
Utility Menu>Select>Everything Below>Selected Keypoints
Utility Menu>Select>Everything Below>Selected Lines
Utility Menu>Select>Everything Below>Selected Volumes

ALPFILL, *LN1*, *LN2*, *LN3*, *LN4*, *LN5*, *LN6*, *LN7*, *LN8*, *LN9*, *LN10*

Fills in an area loop within an existing 2-D area (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

LN1, *LN2*, *LN3*, *LN4*, *LN5*, *LN6*, *LN7*, *LN8*, *LN9*, *LN10*
List of lines that define the loop. If *LN1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). If *LN1* = ALL, all selected lines will be checked for possible closure.

Notes

If all of the lines in the list are not continuous or closed, ANSYS will automatically find the subset of continuous lines within the list.

Use this command to fill in small area loops in models imported from CAD files (this is a geometry "cleanup" tool). This tool is available only for models imported from CAD files (Default IGES option).

Menu Paths

Main Menu>Preprocessor>Modeling>Simplify>Toolkit>Fill Loops

ALPHAD, *VALUE*

Defines the mass matrix multiplier for damping.

SOLUTION: Dynamic Options

MP ME ST <> <> <> <> <> <> PP ED

VALUE
Mass matrix multiplier for damping.

Notes

Defines the mass matrix multiplier, α, for damping. One form of the viscous damping matrix [C] is given by α[M] + β[K], where [M] is the mass matrix and [K] is the stiffness matrix. Damping is not used in the static (**ANTYPE**,STATIC) or buckling (**ANTYPE**,BUCKLE) analyses.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping
Main Menu>Solution>Analysis Type>Sol'n Controls>Transient
Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping

AMAP, *AREA*, *KP1*, *KP2*, *KP3*, *KP4*

Generates a 2-D mapped mesh based on specified area corners.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

AREA
Area number of area to be meshed. If *AREA* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

KP1, *KP2*, *KP3*, *KP4*
Keypoints defining corners of the mapped mesh. Three or four corners may be specified, and may be input in any order.

Notes

Only one area at a time can be meshed with this command. The program *internally* concatenates all lines between the specified keypoints, then meshes the area with all quadrilateral elements. If line divisions are set, the mesh will follow the rules for mapped meshing (see Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide*).

If the area being meshed has concatenated lines, the program will ask if those concatenations should be removed (in batch, the concatenations will automatically be removed). Nodes required for the generated elements are created and assigned the lowest available node numbers. If a mapped mesh is not possible due to mismatched line divisions or poor element shapes, the meshing operation is aborted.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>By Corners

AMESH, *NA1*, *NA2*, *NINC*

Generates nodes and area elements within areas.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NINC*

Mesh areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and all selected areas [**ASEL**] are meshed. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

Notes

Any undefined nodes required for the generated elements are created and assigned the lowest available numbers.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Areas>Free
Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>3 or 4 sided
Main Menu>Preprocessor>Meshing>Mesh>Areas>Target Surf

/AN3D, *Kywrd*, *KEY*

Specifies 3-D annotation functions

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

If *Kywrd* = ANUM, the type of annotation is defined, along with its location. The command format is **/AN3D**,ANUM,*NUM*,*TYPE*,*XHOT*,*YHOT*,*ZHOT*.

NUM

Unique number assigned as each annotation is applied to a model. These numbers are applied sequentially, although when an annotation entity is deleted, its number is reassigned.

TYPE

Annotation internal type number (101 = text, 102 = line, 103 = point, 104 = area, 105 = arrow, 106 = symbol, 108 = bitmap).

XHOT, *YHOT*, *ZHOT*

X, Y, Z coordinates for hot spot location.

If *Kywrd* = BITM, the annotation is a bitmap. The command format is **/AN3D**,BITM,*TYPE*,*X*,*Y*,*Z*.

TYPE
An integer value between 1 and 99, indicating a texture or bitmap. Numbers 1 through 40 correspond to existing ANSYS textures (see **/TXTRE**). Numbers 51 through 99 correspond to textures defined using the File option of the **/TXTRE** command. You can use this capability to override the predefined logo, clamp and arrow files available from the GUI dialog box (numbers 51 through 57). Numbers 41 through 50 are reserved.

X, *Y*, *Z*
X, Y, Z coordinates for the lower left corner of the bitmap.

If *Kywrd* = TEXT, an annotation text string is created. The command format is **/AN3D**,TEXT,*X*,*Y*,*Z*,*Text_String*.

X, *Y*, *Z*
X, Y, Z coordinate location for text string.

Text_String
Text string to be applied as annotation.

If *Kywrd* = LINE, an annotation line is created. The command format is **/AN3D**,LINE,*X1*,*Y1*,*Z1*,*X2*,*Y2*,*Z2*.

X1, *Y1*, *Z1*
X, Y, Z coordinates for beginning of line.

X2, *Y2*, *Z2*
X, Y, Z coordinates for end of line.

If *Kywrd* = POINT, a dot will be drawn in space (this option is not available from the GUI). The command format is **/AN3D**,POINT,*X*,*Y*,*Z*.

X, *Y*, *Z*
X, Y, Z coordinates for point.

If *Kywrd* = AREA, a polygonal area with *n* vertices will be drawn. The command format is **/AN3D**,AREA,*NVERT*,*Xn*,*Yn*,*Zn*.

NVERT
The number of vertices (*n*) for the polygon. Your Polygon can have between 3 and 9 vertices.

Xn, *Yn*, *Zn*
X, Y, Z coordinate location for vertex *n*.

If *Kywrd* = ARROW, an annotation arrow is created. The command format is **/AN3D**,ARROW,*SIZEX1*,*Y1*,*Z1*,*X2*,*Y2*,*Z2*.

SIZE
Symbol size multiplier (0.1 to 20.0, default = 1.0) for the head of the arrow.

X1, *Y1*, *Z1*
X, Y, Z coordinates for the location of the tail.

X2, *Y2*, *Z2*
X, Y, Z coordinate for the location of the tip.

If *Kywrd* = SYMBOL, an annotation symbol is created. The command format is **/AN3D**,SYMBOL,*TYPE*,*X*,*Y*,*Z*,*SIZE*.

TYPE
The symbol type (1 = CAP, 2 = TEE, 3 = CIRCLE, 4 = TRIANGLE, 5 = STAR).

X, *Y*, *Z*
X, Y, Z coordinate location for the symbol.

SIZE
Size multiplier for the symbol (0.1 to 20.0, default = 1.0)

Notes

Because 3-D annotation is applied in relation to the XYZ coordinates of the anchor, you can transform your model, and the annotation will maintain the spatial relationship with the model. This works within reason, and there are instances where changing the perspective or the size of the model will change the apparent relationship between the annotation and the model.

The overall 3-D dimensions of your model are defined by a bounding box. If portions of your model's bounding box lie outside of the visible area of your graphics window (if you are zoomed in on a specific area of your model), it can affect the placement of your 3-D annotations. Zooming out will usually overcome this problem.

3-D annotation is valid for the Cartesian (**CSYS**,0) coordinate system only. If you want to annotate a model you created in another coordinate system, use 2-D annotation (note that 2-D annotations do not remain anchored for dynamic rotations or transformations).

When you apply user defined bitmaps, the size of the annotation can vary. Use the options menu of the 3-D annotation widget to adjust the size and placement of your bitmaps.

You cannot use the "!" and "$" characters in ANSYS text annotation.

The GUI generates this command during 3-D annotation operations and inserts the command into the log file (Jobname.LOG). You should NOT type this command directly during an ANSYS session (although the command can be included in an input file for batch input or for use with the **/INPUT** command).

Menu Paths

Utilty Menu>PlotCtrls>Annotate>Create 3D Annotation

ANCNTR, *NFRAM*, *DELAY*, *NCYCL*

Produces an animated sequence of a contoured deformed shape.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

NFRAM
Number of frames captures (defaults to 5).

DELAY
Time delay during animation (defaults to 0.1 seconds).

NCYCL
Number of animation cycles (defaults to 5). Available in non-UI mode only.

Notes

ANCNTR involves an ANSYS macro which produces an animation of a contoured deformed shape of the last plot action command. This command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANCNTR**, you can replay the animated sequence by issuing the **ANIM** command.

The command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Deformed Results

ANCUT, *NFRAM*, *DELAY*, *NCYCL*, *QOFF*, *KTOP*, *TOPOFF*, *NODE1*, *NODE2*, *NODE3*

Produces an animated sequence of Q-slices.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

NFRAM
Number of frames captures (defaults to 5).

DELAY
Time delay during animation (defaults to 0.1 seconds).

NCYCL
Number of animation cycles (defaults to 5). Available in non-UI mode only.

QOFF
Q-slice working plane increment (defaults to .1 half screens).

KTOP
Topological effect on or off (YES or NO; default is NO).

TOPOFF
Topological offset (default is .1 half screens).

NODE1
Node 1 for start of the Q-slice.

NODE2
Node 2 for direction of the Q-slice.

NODE3
Node 3 for plane of the Q-slice.

Notes

ANCUT involves an ANSYS macro which produces an animation of Q-slices of the last plot action command. This command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANCUT**, you can replay the animated sequence by issuing the **ANIM** command.

The command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Q-Slice Contours
Utility Menu>PlotCtrls>Animate>Q-Slice Vectors

ANCYC, *NUMFRAMES*, *KCYCL*, *DELAY*

Applies a traveling wave animation to graphics data in a modal cyclic symmetry analysis.

POST1: Animation

MP ME ST <> <> <> <> <> <> <> ED

NUMFRAMES
The number of plot frames for the animation. Valid values range from 5 through 36. The default is 18. A low value (because it specifies fewer graphical frames) produces a rougher animation but loads faster. A high value produces a smoother animation but requires more time to load.

KCYCL
The animation mode:

0
Discontinuous animation cycle (forward-reset-forward). This option is the default.

1
Continuous animation cycle (forward-reverse-forward).

DELAY
The time delay (in seconds) between animation frames. Valid values range from 0.1 through 1.0. The default is 0.1 seconds, which produces a seemingly real-time animation. A higher value produces a slower animation.

Command Default

The default **ANCYC** command (issuing the command with no arguments) specifies these implicit argument values: **ANCYC**, 18, 0, 0.1

Notes

The **ANCYC** command is valid in a modal cyclic symmetry analysis only.

The command animates the cyclic symmetry mode shape plot in the General Post Processor (**/POST1**). When you issue a nodal- or element-results plot command (for example, **PLNSOL**, **PLESOL**, or **PLDISP**) and then issue the **ANCYC** command, ANSYS applies a traveling wave animation to the mode shape plot.

Each frame of the animation is created by expanding the cyclic symmetry mode shape at increasing phase angles (via the **/CYCEXPAND** command) starting at zero in equal increments over 360°. The phase-angle increment is 360 / *NUMFRAMES*.

The animation display shows the traveling wave of the result quantity being plotted. The traveling wave animation is applicable only to nodal diameters (harmonic indices) greater than 0 and less than $N/2$ (where N is the number of cyclic sectors in the model).

For more information, see Applying a Traveling Wave Animation to the Cyclic Model in the *ANSYS Advanced Analysis Techniques Guide*.

Menu Paths

Utility Menu> PlotCtrls> Animate> Cyc Traveling Wave

ANDATA, *DELAY*, *NCYCL*, *RSLTDAT*, *MIN*, *MAX*, *INCR*, *FRCLST*, *AUTOCNTRKY*

Produces a sequential contour animation over a range of results data.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

DELAY
Time delay during animation (defaults to 0.5 seconds).

NCYCL
Number of animation cycles (defaults to 5). Available in non-UI mode only.

RSLTDAT
The type of results data to be used for the animation sequence. This can be:

0
Current load step data (default).

1
Range of load step data.

2
Range of results data.

MIN
The range minimum value. If left blank or 0, defaults to the first data point.

MAX
The range maximum value. If left blank or 0, defaults to the last data point.

INCR
The increment between result data (defaults to 1).

FRCLST
Key to force the last sub step in a selected load step to be included in the animation (defaults to 0).

AUTOCNTRKY
Auto-scales contour values, based on the overall subset range of values. The auto-scaling option defaults to 0, no auto-scaling.

Notes

The **ANDATA** command operates only on graphic display platforms supporting the **/SEG** command. It uses an ANSYS macro to produce an animation based on the last plot action command (e.g., **PLDISP**).

The **ANDATA** command implicitly issues **/DSCALE**, 1 for default displacement scaling. Large displacements may not give good results.

This command functions only in the postprocessor.

Menu Paths

Main Menu>Drop Test>Animate Results
Utility Menu>PlotCtrls>Animate>Animate Over Results

ANDSCL, *NFRAM*, *DELAY*, *NCYCL*

Produces an animated sequence of a deformed shape.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

NFRAM
Number of frames captured (defaults to 5).

DELAY
Time delay during animation (defaults to 0.1 seconds).

NCYCL
Number of animation cycles (defaults to 5). Available in non-UI mode only.

Notes

ANDSCL involves an ANSYS macro which produces an animation of displacement of the last plot action command (for example, **PLDISP**). This command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANDSCL**, you can replay the animated sequence by issuing the **ANIM** command.

The command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Deformed Shape

ANDYNA, *DELAY*, *NCYCL*, *START*, *END*, *INC*, *AUTOCONTOURKEY*

Produces an animated sequence of contour values through substeps.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

DELAY
Time delay during animation (defaults to 0.1 seconds).

NCYCL
Number of animation cycles (defaults to 5). Available in non-UI mode only.

START
Number of the starting substep (defaults to 1).

END
Number of the ending substep (defaults to the maximum substep).

INC
Increment between substeps (defaults to 1).

AUTOCONTOURKEY
Auto-scales contour values, based on the overall subset range of values (defaults to 0, no auto-scaling).

Notes

ANDYNA involves an ANSYS macro which produces an animation of contour values through all the substeps of the last plot action command. This command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANDYNA**, you can replay the animated sequence by issuing the **ANIM** command.

The command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Dynamic Results

/ANFILE, *LAB*, *Fname*, *Ext*, --

Saves or resumes an animation sequence to or from a file.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

LAB
Label type.

SAVE
Save the current animation to a file.

RESUME
Resume an animation from a file.

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to ANIM if *Fname* is blank.

--
Unused field

Notes

This command saves an animation to a file from local terminal segments or resumes an animation from a file to local terminal segments. See the **/SEG** command for details on segment storage. See the **ANCNTR** macro for a convenient method of storing graphics frames in terminal memory segments. This command is device dependent and is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Restore Animation
Utility Menu>PlotCtrls>Animate>Save Animation

ANFLOW, *NFRAM*, *DELAY*, *NCYCL*, *TIME*, *SPACING*, *SIZE*, *LENGTH*

Produces an animated sequence of particle flow in a flowing fluid or a charged particle traveling in an electric or magnetic field.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

NFRAM
Number of frames captured (defaults to 5).

DELAY
Time delay during animation (defaults to 0.1 seconds).

NCYCL
Number of animation cycles (defaults to 5). Non-UI mode only.

TIME
Total Trace Time (seconds) (defaults to 0, which is the full flow trace).

SPACING
Particle spacing in seconds (defaults to 0).

SIZE
Particle size (defaults to 0, which is a line).

LENGTH
Particle length fraction (defaults to .1).

Notes

ANFLOW invokes an ANSYS macro which produces an animation of particle flow in a flowing fluid or charged particle motion in an electric or magnetic field by the last plot action command (i.e., **PLTRAC**). This command is only operational on graphic display platforms supporting the **/SEG** command. After executing **ANFLOW**, you can replay the animated sequence by issuing the **ANIM** command. This command is functional only in the Postprocessor.

The *TIME* option lets you set the time interval of forward travel for the trace. The *SPACING* option is used to define the particle spacing in seconds from adjacent particles in the stream line. The *SIZE* variable sets the radius of the particle. The *LENGTH* variable is used to define the particle length fraction. By default, the *LENGTH* is set to .1, which means the particle occupies 10% of the flow region and the other 90% is a color-code line. The *SPACING* and *LENGTH* variables only make sense when the *SIZE* variable is nonzero (i.e., the particle is bigger than the line).

Menu Paths

Utility Menu>PlotCtrls>Animate>Particle Flow

/ANGLE, *WN*, *THETA*, *Axis*, *KINCR*

Rotates the display about an axis.

GRAPHICS: Views

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

THETA
Angle (degrees) for changing display orientation (positive, counterclockwise about specified axis).

Axis
Rotation axis: XS, YS, or ZS (default) for the screen axes; XM, YM, or ZM for the global Cartesian model axes. ZS is normal to the screen; all axes pass through the focus point.

KINCR
Cumulative rotation key:

0
Do not use cumulative successive rotations.

1
Use cumulative rotations. Rotations are relative to the previous rotation. View settings (**/VIEW**) are recalculated.

Notes

Default orientation is YS vertical. When the **/XFRM** command is set for rotation about two points, or for entities, the **/ANGLE** command is functional only for *Axis* = ZS or ZM and *KINCR* = 1.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate
Utility Menu>PlotCtrls>View Settings>Angle of Rotation

ANHARM, *NFRAM*, *DELAY*, *NCYCL*

Produces a time-transient animated sequence of time-harmonic results (ANTYPE,HARMIC).

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

NFRAM
Number of frame captures per cycle. Defaults to 12.

DELAY
Time delay during animation. Defaults to 0.1 seconds.

NCYCL
Number of animation cycles. Defaults to 5. Not available in the GUI.

Notes

ANHARM invokes an ANSYS macro which produces a time-transient animation of time-harmonic results of the last plot action command (e.g. **PLNSOL**,B,SUM). The animation converts the complex solution variables (real and imaginary sets) into time varying results over one period. For example, if *NFRAM* = 12, then the frame captures are in increments of 30 degree phase angles.

Menu Paths

Utility Menu>PlotCtrls>Animate>Animate Over Time
Utility Menu>PlotCtrls>Animate>Time-harmonic

ANIM, *NCYCL*, *KCYCL*, *DELAY*

Displays graphics data in animated form.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

NCYCL
Number of cycles associated with the animation (defaults to 5 in non-GUI mode only)

KCYCL
Animation mode:

0
Continuous animation cycle (forward-reverse-forward-etc.) (default).

1
Discontinuous animation cycle (forward-reset-forward-etc.).

DELAY
Time delay (seconds) between animation frames (defaults to 0.1 seconds).

Notes

Displays graphics data stored in local terminal segments in animated form. See the **/SEG** command for details on segment storage. See the **ANCNTR** macro for a convenient method of storing graphics frames in terminal memory segments. This command is device-dependent. You should not resize the graphic while animation is in progress; doing so can result in distorted plots.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Replay Animation
Utility Menu>PlotCtrls>Animate>Restore Animation

ANISOS, *NFRAM*, *DELAY*, *NCYCL*

Produces an animated sequence of an isosurface.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

NFRAM
Number of frames captures (defaults to 9).

DELAY
Time delay during animation (defaults to 0.1 seconds).

NCYCL
Number of animation cycles (defaults to 5). Available in non-UI mode only.

Notes

ANISOS involves an ANSYS macro which produces an animation of an isosurface of the last plot action command (for example, **PLNSOL**,S,EQV). The **ANISOS** command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANISOS**, you can replay the animated sequence by issuing the **ANIM** command.

This command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Isosurfaces

ANMODE, *NFRAM*, *DELAY*, *NCYCL*, *KACCEL*

Produces an animated sequence of a mode shape.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

NFRAM
Number of frames captures (defaults to 5).

DELAY
Time delay during animation (defaults to 0.1 seconds).

NCYCL
Number of animation cycles (defaults to 5). Available in non-UI mode only.

KACCEL
Acceleration type:

0
Linear acceleration.

1
Sinusoidal acceleration.

Notes

ANMODE involves an ANSYS macro which produces an animation of mode shape of the last plot action command (for example, **PLDISP**). The **ANMODE** command operates only on graphic display platforms supporting the **/SEG** command. After executing **ANMODE**, you can replay the animated sequence by issuing the **ANIM** command.

This command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Mode Shape

ANMRES*,DELAY, MIN, MAX, INC, AUTOCNTRKY, FREQ, EXT*

Performs animation of results over multiple results files in an explicit dynamic structural analysis or fluid flow analysis with remeshing.

POST1: Animation

<> <> <> DY <> <> <> <> <> <> ED

DELAY
Time delay during animation (default = 0.5 seconds).

MIN
Minimum results file number to animate. Default = 1 (for **Jobname.RS01**).

MAX
Maximum results file number to animate. Defaults to the highest numbered results file, **Jobname.RSnn**.

INC
Increment between results file numbers. Default = 1.

AUTOCNTRKY
Automatic contour scaling option.

0
No auto-scaling (default).

1
Auto-scaling on.

If you activate automatic contour scaling, ANSYS considers only the minimum and maximum value of the result item from the first results file.

FREQ
Results frequency key.

0 or 1
Animate every results set in each **Jobname.EXT** file (default).

2
Animate every other results set in each **Jobname.EXT** file.

n
Animate every *n*th results set in each **Jobname.EXT** file.

EXT
Extension of result files

'rfl'
Animate **Jobname.rflnn**

'rs'
Animate **Jobname.rsnn**. Default = 'rs'.

Notes

ANMRES invokes an ANSYS macro that performs animation across multiple results files (**Jobname.EXT**, **Jobname.EXT**, etc.) produced by an explicit dynamic structural analysis or fluid flow analysis with remeshing. Multiple results files typically occur when adaptive meshing is used in an explicit dynamic structural analysis or fluid flow analysis with remeshing. **ANMRES** cannot be used for multiple results files that are caused by file splitting.

ANMRES animates results from files having the currently specified jobname (**Jobname.EXT** - **Jobname.EXT**). To change the current jobname, use the **/FILNAME** command. The animation is based on the last plot command (e.g., **PLDISP**).

Menu Paths

Main Menu>Drop Test>Animate Results
Utility Menu>PlotCtrls>Animate>Animate Over Results

/ANNOT, *Lab*, *VAL1*, *VAL2*

Activates graphics for annotating displays (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

Lab
Annotation control key:

OFF
Turns off annotation for each subsequent display (default).

ON
Turns on annotation for each subsequent display.

DELE
Deletes all annotation.

SAVE
Saves annotation on a file. Use *VAL1* for file name (defaults to **Jobname**) and *VAL2* for the extension (defaults to ANO).

SCALE
Sets annotation scale factor (direct input only). Use *VAL1* for value (0.1 to 10.0) (defaults to 1.0).

XORIG
Sets the annotation x origin (direct input only). Use *VAL1* for value (-3.0 to 3.0).

YORIG
Sets annotation y origin (direct input only). Use *VAL1* for value (-3.0 to 3.0).

SNAP
Sets annotation snap (menu button input only). Use *VAL1* for value (0.002 to 0.2) (defaults to 0.002).

STAT
Displays current annotation status.

DEFA
Sets annotation specifications to the default values.

REFR
Redisplays annotation graphics.

TMOD
Sets the annotation text mode. If *VAL1* = 1, annotation text will be drawn in scalable bitmap fonts (default). If *VAL1* = 0, annotation text will be drawn with stroke text.

VAL1
Value (or file name) as noted with label above.

VAL2
Value (or file name extension) as noted with label above.

Notes

This is a command generated by the GUI and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

You cannot use the "!" and "$" characters in ANSYS text annotation.

/ANNOT activates annotation graphics for adding annotation to displays. Commands representing the annotation instructions are automatically created by the annotation functions in the GUI and written to **Jobname.LOG**. The annotation commands are **/ANNOT**, **/ANUM**, **/TLABEL**, **/LINE**, **/LARC**, **/LSYMBOL**, **/POLYGON**, **/PMORE**, **/PCIRCLE**, **/PWEDGE**, **/TSPEC**, **/LSPEC**, and **/PSPEC**. Annotation graphics are relative to the full Graphics Window and are not affected by ANSYS window-specific commands (**/WINDOW**, **/VIEW**, etc.).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

ANORM, *ANUM*, *NOEFLIP*

Reorients area normals.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

ANUM
Area number having the normal direction that the reoriented areas are to match.

NOEFLIP
Indicates whether you want to change the normal direction of the existing elements on the reoriented area(s) so that they are consistent with each area's new normal direction.

0
Make the normal direction of existing elements on the reoriented area(s) consistent with each area's new normal direction (default).

1
Do not change the normal direction of existing elements on the reoriented area(s).

Notes

Reorients areas so that their normals are consistent with that of a specified area.

If any of the areas have inner loops, the **ANORM** command will consider the inner loops when it reorients the area normals.

You cannot use the **ANORM** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal.

See Revising Your Model of the *ANSYS Modeling and Meshing Guide* for more information.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Areas>Area Normals

---

ANSOL, *NVAR*, *NODE*, *Item*, *Comp*, *Name*, *Mat*, *Real*, *Ename*

Specifies averaged nodal data to be stored from the results file in the solution coordinate system.

POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

NVAR
Arbitrary reference number assigned to this variable (2 to *NV* [**NUMVAR**]). Overwrites any existing results for this variable.

NODE
Node number for which data are to be stored.

Item
Label identifying the item. General item labels are shown in ANSOL - General Item and Component Labels below. Some items also require a component label.

Comp
Component of the item (if required). General component labels are shown in ANSOL - General Item and Component Labels below.

Name
Thirty-two character name for identifying the item on the printout and displays. Defaults to an eight character label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Mat
The material number. Average will be computed based on the subset of elements with the specified material number. DEFAULT: Use all elements in the active set unless *Real* and/or *Ename* is specified.

Real
The real number. Average will be computed based on the subset of elements with the specified real number. DEFAULT: Use all elements in the active set unless *Mat* and/or *Ename* is specified.

Ename
The element type name. Average will be computed based on the subset of elements with the specified element type name. DEFAULT: Use all elements in the active set unless *Mat* and/or *Real* is specified.

Notes

Valid item and component labels for averaged nodal results are listed in ANSOL - General Item and Component Labels, below.

All element nodal quantities are obtained in **RSYS**, Solu and then averaged.

The **ANSOL** command defines averaged nodal results data to be stored from a results file [**FILE**]. Not all items are valid for all nodes. See the input and output summary tables of the *ANSYS Elements Reference* of each element that is attached to the node for the available items.

COORDINATE SYSTEMS: All element nodal results used by **ANSOL** for averaging are in the element coordinate system, except for layered elements. Layered element results are in the layer coordinate system. You can further specify the element nodal results, for some elements, with the **SHELL**, **LAYERP26**, and **FORCE** commands.

ANSOL does not transform results from **RSYS**, SOLU to other coordinate systems. Verify that all elements attached to the subject node have the same coordinate system before using **ANSOL**.

SHELL ELEMENTS: The default shell element coordinate system is based on node ordering. For shell elements the adjacent elements could have a different **RSYS**,SOLU, making the resultant averaged data inconsistent. A note to this effect is issued when **ANSOL** is used in models containing shell elements. Ensure that consistent coordinate systems are active for all associated elements used by the **ANSOL** command.

DERIVED QUANTITIES: Some of the result items supported by **ANSOL** (see ANSOL - General Item and Component Labels) are derived from the component quantities. Use **AVPRIN** to specify the principal and vector sum quantity averaging methods.

DEFAULT: If *Mat*, *Real* , and *Ename* are not specified, all of the elements attached to the node will be considered. When a material ID, real constant ID, or element type discontinuity is detected at a node, a note is issued. For example, in a FSI analysis, a FLUID30 element at the structure interface would be considered. But since it contains no *SX* result, it will not be used during **STORE** operations.

ANSOL - General Item and Component Labels

General Item and Component Labels ANSOL, *NVAR, ELEM, NODE, Item, Comp, Name, Mat, Real, Ename*

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.

General Item and Component Labels ANSOL, *NVAR, ELEM, NODE, Item, Comp, Name, Mat, Real, Ename*

Item	Comp	Description
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1,2,3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
CONT	STAT	Contact status: 3-closed and sticking 2-closed and sliding 1-open but near contact 0-open and not near contact
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress
"	STOT	Contact total stress (pressure plus friction)
"	SLIDE	Contact sliding distance
"	GAP	Contact gap distance
"	FLUX	Total heat flux at contact surface
"	CNOS	Total number of contact status changes during substep.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.

General Item and Component Labels ANSOL, *NVAR, ELEM, NODE, Item, Comp, Name, Mat, Real, Ename*

Item	Comp	Description
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).

Menu Paths

Main Menu>TimeHist Postpro>Variable Viewer

ANTIME, *NFRAM*, *DELAY*, *NCYCL*, *AUTOCNTRKY*, *RSLTDAT*, *MIN*, *MAX*

Produces a sequential contour animation over a range of time.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

NFRAM
Number of frame captures (defaults to 5).

DELAY
Time delay during animation (defaults to 0.1 seconds).

NCYCL
Number of animation cycles (defaults to 5). Available in non-UI mode only.

AUTOCNTRKY
Auto-scales contour values, based on the overall subset range of values. The auto-scaling option defaults to 0, no auto-scaling.

RSLTDAT
The results data to be used for the animation sequence. This can be:

0
Current load step data (default).

1
Range of load step data.

2
Range of time data.

MIN
The range minimum value. If left blank defaults to the first data point.

MAX
The range maximum value. If left blank defaults to the last data point.

Notes

The **ANTIME** command operates only on graphic display platforms supporting the **/SEG** command. It uses an ANSYS macro to produce an animation of contour values for the last plot action command (for example, **PLDISP**). After executing **ANTIME**, the **ANIM** command will replay the animated sequence.

This command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Animate Over Time
Utility Menu>PlotCtrls>Animate>Time-harmonic

ANTYPE, *Antype*, *Status*, *LDSTEP*, *SUBSTEP*, *Action*

Specifies the analysis type and restart status.

SOLUTION: Analysis Options

MP ME ST <> <> PR EM EH <> PP ED

Antype

Analysis type (defaults to the previously specified analysis type, or to STATIC if none specified):

STATIC or 0
: Perform a static analysis. Valid for all degrees of freedom.

BUCKLE or 1
: Perform a buckling analysis. Implies that a previous static solution was performed with prestress effects calculated [**PSTRES**,ON]. Valid for structural degrees of freedom only.

MODAL or 2
: Perform a modal analysis. Valid for structural and fluid degrees of freedom.

HARMIC or 3
: Perform a harmonic analysis. Valid for structural, fluid, magnetic, and electrical degrees of freedom.

TRANS or 4
: Perform a transient analysis. Valid for all degrees of freedom.

SUBSTR or 7
: Perform a substructure analysis. Valid for all degrees of freedom.

SPECTR or 8
: Perform a spectrum analysis. Implies that a previous modal analysis was performed. Valid for structural degrees of freedom only.

Status

Specifies the status of the analysis (new or restart):

NEW
: Specifies a new analysis (default). If NEW, the remaining fields on this command are ignored.

REST
: Specifies a restart of a previous analysis. Valid only for static, harmonic (2-D magnetic only), full transient, and substructure analyses (backsubstitution method only). For full transient and nonlinear static structural analyses, a multiframe restart will be done by default. Use the **RESCONTROL** command to set up or disable the multiframe restart.

  This option resumes the **.rdb** file created at the start of solution. If boundary conditions are deleted in solution (for example, after being used to create an initial velocity or to establish initial contact), they will need to be deleted again after issuing this command.

LDSTEP
Specifies the load step at which a multiframe restart will begin. The default is the highest load step number found in the **Jobname.R*nnn*** files for the current jobname in the current directory. Not used for single frame restarts.

SUBSTEP
Specifies the substep at which a multiframe restart will begin. The default is the highest substep number found for the specified *LDSTEP* in the **Jobname.R*nnn*** files in the current directory. Not used for single frame restarts.

Action
Specifies the manner of a multiframe restart. Not used for traditional restarts.

CONTINUE
ANSYS will continue the analysis based on the specified *LDSTEP* and *SUBSTEP* (default). The current load step will be continued unless the end of the load step is encountered in the **.R*nnn*** file, in which case a new load step will be started. ANSYS will delete all **.R*nnn*** files beyond the point of restart and will update the **.LDHI** file if a new load step is encountered.

ENDSTEP
At restart, force the specified load step (LDSTEP) to end at the specified substep (SUBSTEP), even though the end of the current load step has not been reached. At the end of the specified substep, all loadings will be scaled to the level of the current ending and stored in the **.LDHI** file. A run following this ENDSTEP will start a new load step. This feature allows you to change the load level in the middle of a load step. ANSYS will update the **.LDHI** file and delete all **.R*nnn*** files beyond the point of ENDSTEP. The **.R*nnn*** file at the point of ENDSTEP will be rewritten to record the rescaled load level.

RSTCREATE
At restart, retrieve information to be written to the results file for the specified load step (LDSTEP) and substep (SUBSTEP). Be sure to use **OUTRES** to write the results to the **.RST** file. This action does not affect the **.LDHI** or **.R*nnn*** files. Previous items stored in the **.RST** file at and beyond the point of RSTCREATE will be deleted.

Command Default

New static analysis.

Notes

The analysis type *Antype* cannot be changed if it is a restart run. Always save parameters before doing a restart. You can do a multiframe restart only for nonlinear static and full transient structural analyses. (For linear static analyses, you can use a single frame restart.)

The following notes apply to multiframe restart:

- *LDSTEP*, *SUBSTEP*, and *Action* are only valid for multiframe restart.
- To switch from multiframe restart to the single frame restart, delete any **.RDB**, **.LDHI**, and **.R*nnn*** files in the directory. Issue the **ANTYPE**,,REST command and use the files **.DB**, **.ESAV** or **.OSAV**, and **.EMAT**.
- A new specification of the **TIME** command within a load step when beginning a multiframe restart is invalid because the load step has been specified previously before restart.

This command is also valid in PREP7.

Product Restrictions

Only the *Antype* values STATIC, BUCKLE, MODAL, HARMIC, TRANS, or SPECTR are valid in ANSYS Professional. Only the *Antype* values STATIC, HARMIC, or TRANS are valid in ANSYS Emag.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>New Analysis
Main Menu>Preprocessor>Loads>Analysis Type>Restart
Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic
Main Menu>Solution>Analysis Type>New Analysis
Main Menu>Solution>Analysis Type>Restart
Main Menu>Solution>Analysis Type>Sol'n Controls>Basic

/ANUM, *NUM*, *TYPE*, *XHOT*, *YHOT*

Specifies the annotation number, type, and hot spot (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

NUM
Annotation number. ANSYS automatically assigns the lowest available number. You cannot assign a higher number if a lower number is available; ANSYS will substitute the lowest available number in place of any user-specified higher number.

TYPE
Annotation internal type number. If *TYPE* = DELE, delete annotation *NUM*.

1
Text

2
Block text (not available in GUI)

3
Dimensions

4
Lines

5
Rectangles

6
Circles

7
Polygons

8
Arcs

9
Wedges, pies

11
Symbols

12
Arrows

13
Bitmap

XHOT
X hot spot (-1.0 < X < 2.0). Used for menu button item delete.

YHOT
Y hot spot (-1.0 < Y < 1.0). Used for menu button item delete.

Command Default

Number, type, and hot spot are automatically determined.

Notes

This is a command generated by the GUI and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

Type 13 (bitmap) annotation applies user defined bitmaps defined using the FILE option of the **/TXTRE** command.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

AOFFST, *NAREA*, *DIST*, *KINC*
Generates an area, offset from a given area.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NAREA
Area from which generated area is to be offset. If *NAREA* = ALL, offset from all selected areas [**ASEL**]. If *NAREA* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

DIST
Distance normal to given area at which keypoints for generated area are to be located. Positive normal is determined from the right-hand-rule keypoint order.

KINC
Keypoint increment between areas. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

Notes

Generates an area (and its corresponding keypoints and lines) offset from a given area. The direction of the offset varies with the given area normal. End slopes of the generated lines remain the same as those of the given pattern. Area and line numbers are automatically assigned, beginning with the lowest available values [**NUMSTR**].

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Offset

---

AOVLAP, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Overlaps areas.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Numbers of areas to be operated on. If *NA1* = ALL, use all selected areas and ignore *NA2* to *NA9*. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

Notes

Generates new areas which encompass the geometry of all the input areas. The new areas are defined by the regions of intersection of the input areas, and by the complementary (non-intersecting) regions. See Solid Modeling in the *ANSYS Modeling and Meshing Guide* for an illustration. This operation is only valid when the region of intersection is an area. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Overlap>Areas

---

APLOT, *NA1*, *NA2*, *NINC*, *DEGEN*, *SCALE*

Displays the selected areas.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NINC*

Displays areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL (default), *NA2* and *NINC* are ignored and all selected areas [**ASEL**] are displayed.

DEGEN

Degeneracy marker:

(blank)

No degeneracy marker is used (default).

DEGE
A red star is placed on keypoints at degeneracies (see the *ANSYS Modeling and Meshing Guide*). Not available if **/FACET**,WIRE is set.

SCALE
Scale factor for the size of the degeneracy-marker star. The scale is the size in window space (-1 to 1 in both directions) (defaults to .075).

Notes

This command is valid in any processor. The degree of tessellation used to plot the selected areas is set through the **/FACET** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>Plot Degen Areas
Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>Plot Degen Areas
Utility Menu>Plot>Areas
Utility Menu>Plot>Specified Entities>Areas

APPEND, *LSTEP*, *SBSTEP*, *FACT*, *KIMG*, *TIME*, *ANGLE*, *NSET*

Reads data from the results file and appends it to the database.

POST1: Set Up

MP ME ST DY <> PR EM <> FL PP ED

LSTEP
Load step number of the data set to be read. Defaults to 1. If FIRST, ignore *SBSTEP* and *TIME* and read the first data set. If LAST, ignore *SBSTEP* and *TIME* and read the last data set. If NEXT, ignore *SBSTEP* and *TIME* and read the next data set. If already at the last data set, the next set is the first data set. If NEAR, ignore *SBSTEP* and read the data set nearest to *TIME*. If *TIME* is blank, read the first data set. If LIST, scan the results file to produce a summary of each load step (*FACT*, *KIMG*, *TIME* and *ANGLE* are ignored).

SBSTEP
Substep number (within *LSTEP*) (defaults to last substep of load step). For the Buckling (**ANTYPE**,BUCKLE) or Modal (**ANTYPE**,MODAL) analysis, the substep corresponds to the mode number (defaults to first mode). If *LSTEP* = LIST, *SBSTEP* = 0 or 1 will list the basic load step information; *SBSTEP* = 2 will also list the load step title, and label the imaginary data sets if they exist.

FACT
Scale factor applied to data read from the file. If zero (or blank), a value of 1.0 is used. Harmonic velocities or accelerations may be calculated from the displacement results from a Modal or Harmonic Response (**ANTYPE**,HARMIC) analyses. If *FACT* = VELO, the harmonic velocities (v) are calculated from the displacements (d) at a particular frequency (f) according to the relationship $v = 2\pi fd$. Similarly, if *FACT* = ACEL, the harmonic accelerations (a) are calculated as $a = (2\pi f)^2 d$.

KIMG
Used only with results from complex analyses:

0
Store real part of complex solution.

1
Store imaginary part.

TIME
Time-point identifying the data set to be read. For the harmonic response analyses, time corresponds to the frequency. For the buckling analysis, time corresponds to the load factor. Used only in the following cases: If *LSTEP* is NEAR, read the data set nearest to *TIME*. If both *LSTEP* and *SBSTEP* are zero (or blank), read data set at time = *TIME*. If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets. Solution items not written to the results file [**OUTRES**] for either data set will result in a null item after data set interpolation. If *TIME* is beyond the last time point on the file, the last time point is used.

ANGLE
Circumferential location (0° to 360°). Defines the circumferential location for the harmonic calculations used when reading from the results file. The harmonic factor (based on the circumferential angle) is applied to the harmonic elements (PLANE25, PLANE75, PLANE78, FLUID81, PLANE83, and SHELL61) of the load case. See the *ANSYS, Inc. Theory Reference* for details. Note that factored values of applied constraints and loads will overwrite any values existing in the database.

NSET
Data set number of the data set to be read. If a positive value for *NSET* is entered, *LSTEP*, *SBSTEP*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by **APPEND**,LIST. To determine if data sets are real or imaginary, issue **APPEND**,LIST,2 which labels imaginary data sets.

Notes

Reads a data set from the results file and appends it to the existing data in the database for the selected model only. The existing database is not cleared (or overwritten in total), allowing the requested results data to be merged into the database. Various operations may also be performed during the read operation. The database must have the model geometry available (or used the **RESUME** command before the **APPEND** command to restore the geometry from **File.DB**).

Menu Paths

Main Menu>General Postproc>Read Results>By Load Step
Main Menu>General Postproc>Read Results>By Set Number
Main Menu>General Postproc>Read Results>By Time/Freq

APTN, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Partitions areas.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*
Numbers of areas to be operated on. If *NA1* = ALL, *NA2* to *NA9* are ignored and all selected areas are used. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may be substituted for *NA1*.

Notes

Partitions areas that intersect. This command is similar to the combined functionality of the **ASBA** and **AOVLAP** commands. If the intersection of two or more areas is an area (i.e., planar), new areas will be created with boundaries that conform to the area of intersection and to the boundaries of the non-intersecting portions of the input areas [**AOVLAP**]. If the intersection is a line (i.e., not planar), the areas will be subtracted, or divided, along the line(s) of intersection [**ASBA**]. Both types of intersection can occur during a single **APTN** operation. Areas that do not intersect will not be modified. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Partition>Areas

---

ARCLEN, *Key*, *MAXARC*, *MINARC*

Activates the arc-length method.

SOLUTION: Nonlinear Options

MP ME ST <> <> <> <> <> <> PP ED

Key
Arc-length key:

OFF
Do not use the arc-length method (default).

ON
Use the arc-length method.

MAXARC
Maximum multiplier of the reference arc-length radius (default = 25).

MINARC
Minimum multiplier of the reference arc-length radius (default = 1/1000).

Notes

Activates the arc-length method and sets the minimum and maximum multipliers for the arc-length radius. The reference arc-length radius is calculated from the load or displacement increment of the first iteration of the first substep. This increment is determined by the following formula:

Reference Arc-Length Radius = Total Load (or Displacement) / *NSBSTP*

where *NSBSTP* is the number of substeps specified on the **NSUBST** command.

The factors *MAXARC* and *MINARC* are used to define the limits of the arc-length radius by using the following formulas:

lower limit = *MINARC* * (Reference Arc-Length Radius)

upper limit = *MAXARC* * (Reference Arc-Length Radius)

In each subsequent substep, a new arc-length radius is first calculated based on the arc-length radius of the previous substep and the solution behavior. Next, the newly calculated arc-length radius is further modified so that it falls between the range of the upper limit and lower limit. If the solution does not converge even when using the lower limit of the arc-length radius, the solution will terminate.

These values, together with the reference arc-length radius, define the limit for the new arc-length radius.

ARCLEN must be turned OFF for any load step without an applied load or displacement.

You cannot use the arc-length method with the following controls: automatic time stepping [**AUTOTS**], line search [**LNSRCH**], and the DOF solution predictor [**PRED**]. If you activate the arc-length method after you set any of these controls, a warning message appears. If you choose to proceed with the arc-length method activation, ANSYS disables your automatic time stepping, line search, and DOF predictor settings.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Advanced NL
Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Arc-Length Opts
Main Menu>Solution>Analysis Type>Sol'n Controls>Advanced NL
Main Menu>Solution>Load Step Opts>Nonlinear>Arc-Length Opts

ARCOLLAPSE, *AREA*, *LINE*

Collapses specified area to a specified line segment (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

AREA
The ID of the area to collapse.

LINE
The ID of a line belonging to *AREA*; only the specified line will remain after the area is collapsed.

Notes

Use this command to simplify the geometry of a model imported from a CAD file (this is a geometry "cleanup" tool). This tool is available only for models imported from CAD files.

If *AREA* has any attached loads or boundary conditions, these must be reattached after the collapse operation (Default IGES option).

Menu Paths

Main Menu>Preprocessor>Modeling>Simplify>Toolkit>Collapse Areas

---

ARCTRM, *Lab*, *VAL*, *NODE*, *DOF*

Controls termination of the arc-length solution.

SOLUTION: Nonlinear Options

MP ME ST <> <> <> <> <> <> PP ED

Lab

Specifies the basis of solution termination:

OFF

Does not use **ARCTRM** to terminate analysis (default).

L

Terminates the analysis if the first limit point has been reached. The first limit point is that point in the response history when the tangent stiffness matrix becomes singular (i.e., the point at which the structure becomes unstable). If *Lab* = L, arguments *VAL*, *NODE*, *DOF* are ignored.

U

Terminates the analysis when the displacement first equals or exceeds the maximum desired value.

VAL

Maximum desired displacement (absolute value). Valid only if *Lab* = U. The analysis terminates whenever the calculated displacement first equals or exceeds this value. For rotational degrees of freedom, *VAL* must be in radians (not degrees).

NODE

Node number corresponding to displacement used to compare with displacement specified by *VAL*. If blank, the maximum displacement will be used. Valid only if *Lab* = U.

DOF

Valid degree of freedom label for nodal displacement specified by *NODE*. Valid labels are UX, UY, UZ, ROTX, ROTY, ROTZ. Valid only if *NODE*>0 and *Lab* = U.

Notes

It can be convenient to use this command to terminate the analysis when the first limit point is reached. In addition, the **NCNV** command should be used to limit the maximum number of iterations. If the **ARCTRM** command is not used, and the applied load is so large that the solution path can never reach that load, the arc-length solution will continue to run until a CPU time limit or a "maximum number of iterations" is reached.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Advanced NL
Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Arc-Length Opts
Main Menu>Solution>Analysis Type>Sol'n Controls>Advanced NL
Main Menu>Solution>Load Step Opts>Nonlinear>Arc-Length Opts

ARDETACH, *AREA1*, *AREA2*, *AINC*

Detaches areas from neighboring geometrical entities (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

AREA1, *AREA2*, *AINC*

Detach areas from *AREA1* to *AREA2* (defaults to *AREA1*) in steps of *AINC* (defaults to 1).

Notes

Use this command to detach non-manifold areas from their neighboring geometric entities. This command is available only for repairing the geometry of models imported from CAD systems (Default IGES option).

Menu Paths

Main Menu>Preprocessor>Modeling>Geom Repair>Detach Areas

AREAS

Specifies "Areas" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

This command cannot be accessed from a menu.

AREFINE, *NA1*, *NA2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

Refines the mesh around specified areas.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NINC*

Areas (*NA1* to *NA2* in increments of *NINC*) around which the mesh is to be refined. *NA2* defaults to *NA1*, and *NINC* defaults to 1. If *NA1* = ALL, *NA2* and *NINC* are ignored and all selected areas are used for refinement. If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

LEVEL
Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

DEPTH
Depth of mesh refinement in terms of the number of elements outward from the indicated areas (defaults to 1).

POST
Type of postprocessing to be done after element splitting, in order to improve element quality:

OFF
No postprocessing will be done.

SMOOTH
Smoothing will be done. Node locations may change.

CLEAN
Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

RETAIN
Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The ANSYS program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

ON
The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

OFF
The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

Notes

AREFINE performs local mesh refinement around the specified areas. By default, the indicated elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

AREFINE refines all area elements and tetrahedral volume elements that are adjacent to the specified areas. Any volume elements that are adjacent to the specified areas, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes [**IC**], coupled nodes [**CP** family of commands], constraint equations [**CE** family of commands], or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. See Revising Your Model in the *ANSYS Modeling and Meshing Guide* for additional restrictions on mesh refinement.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Areas

AREVERSE, *ANUM*, *NOEFLIP*

Reverses the normal of an area, regardless of its connectivity or mesh status.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

ANUM

Area number of the area whose normal is to be reversed. If *ANUM* = ALL, the normals of all selected areas will be reversed. If *ANUM* = P, graphical picking is enabled. A component name may also be substituted for *ANUM*.

NOEFLIP

Indicates whether you want to change the normal direction of the existing elements on the reversed area(s) so that they are consistent with each area's new normal direction.

0

Make the normal direction of existing elements on the reversed area(s) consistent with each area's new normal direction (default).

1

Do not change the normal direction of existing elements on the reversed area(s).

Notes

You cannot use the **AREVERSE** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable. Also, you cannot use this command to change the normal direction for areas attached to volumes because IGES data is unchanged by reversal. Reversed areas that are attached to volumes need to be reversed again when imported.

Real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal.

See Revising Your Model in the *ANSYS Modeling and Meshing Guide* for more information.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Reverse Normals>of Areas

---

ARFILL, *LN1*, *LN2*, *LN3*, *LN4*, *LN5*, *LN6*, *LN7*, *LN8*, *LN9*, *LN10*

Creates an area based on a set of singly-connected lines (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

LN1, *LN2*, *LN3*, *LN4*, *LN5*, *LN6*, *LN7*, *LN8*, *LN9*, *LN10*

List of lines that define the new area. If *LN1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). If *LN1* = ALL, all selected lines will be checked for possible closure and all open areas will be filled with lines (lines will be added) that form closed loops.

Notes

The **ARFILL** command creates an area based on the boundary defined by a set of singly-connected lines. No lines in the selected set can be connected to two areas. The area created is the minimum surface defined by the boundary line set.

This tool is available only for models imported from CAD files (Default IGES option).

Menu Paths

Main Menu>Preprocessor>Modeling>Geom Repair>Fill Areas

ARMERGE, *A1*, *A2*, *A3*, *A4*, *A5*, *A6*, *A7*, *A8*, *A9*, *A10*

Merges two or more singly-connected adjacent areas (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

A1, *A2*, *A3*, *A4*, *A5*, *A6*, *A7*, *A8*, *A9*, *A10*

List of areas that define the set of areas to merge. If *A1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). If *A1* = ALL, all selected areas will be merged and all remaining arguments are ignored.

Notes

The **ARMERGE** command can concatenate more than two adjacent areas; however, for best results you should limit each merge to two areas. Also, you should restrict merge operations to areas that are clearly simple extensions of each other.

If the areas specified in the list have any attached loads or boundary conditions, these will be removed and must be reattached after the merge operation. If the merge operation would result in abnormal parameterization, the command will fail.

Use this command to simplify the geometry of a model imported from a CAD file (this is a geometry "cleanup" tool). This tool is available only for models imported from CAD files (Default IGES option).

Menu Paths

Main Menu>Preprocessor>Modeling>Simplify>Toolkit>Merge Areas

AROTAT, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *PAX1*, *PAX2*, *ARC*, *NSEG*

Generates cylindrical areas by rotating a line pattern about an axis.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*

List of lines in the pattern to be rotated (6 maximum if using keyboard entry of *NL1* to *NL6*). The lines must lie in the plane of the axis of rotation. If *NL1* = P, graphical picking is enabled and all remaining arguments

are ignored (valid only in the GUI). If *NL1* = ALL, all selected lines will define the pattern to be rotated. A component name may also be substituted for *NL1*.

PAX1, *PAX2*
Keypoints defining the axis about which the line pattern is to be rotated.

ARC
Arc length (in degrees). Positive follows right-hand rule about *PAX1-PAX2* vector. Defaults to 360°.

NSEG
Number of areas (8 maximum) around circumference. Defaults to minimum number required for 90° -maximum arcs, i.e., 4 for 360°, 3 for 270°, etc.

Notes

Generates cylindrical areas (and their corresponding keypoints and lines) by rotating a line pattern (and its associated keypoint pattern) about an axis. Keypoint patterns are generated at regular angular locations, based on a maximum spacing of 90°. Line patterns are generated at the keypoint patterns. Arc lines are also generated to connect the keypoints circumferentially. Keypoint, line, and area numbers are automatically assigned, beginning with the lowest available values [**NUMSTR**]. Adjacent lines use a common keypoint. Adjacent areas use a common line.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Lines>About Axis

---

ARSCALE, *NA1*, *NA2*, *NINC*, *RX*, *RY*, *RZ*, *KINC*, *NOELEM*, *IMOVE*

Generates a scaled set of areas from a pattern of areas.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NA1, *NA2*, *NINC*
Set of areas, *NA1* to *NA2* in steps of *NINC*, that defines the pattern to be scaled. *NA2* defaults to *NA1*, *NINC* defaults to 1. If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is defined by all selected areas. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

RX, *RY*, *RZ*
Scale factors to be applied to the X, Y, and Z keypoint coordinates in the active coordinate system. (*RR*, *R* θ, *RZ* for cylindrical; *RR*, *R* θ, *R* Φ for spherical). Note that the *R* θ and *R* Φ scale factors are interpreted as angular *offsets*. For example, if CSYS = 1, *RX, RY, RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

KINC
Increment to be applied to keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned [**NUMSTR**].

NOELEM
Specifies whether nodes and elements are also to be generated:

0
Nodes and elements associated with the original areas will be generated (scaled) if they exist.

1
Nodes and elements will *not* be generated.

IMOVE
Specifies whether areas will be moved or newly defined:

0
Additional areas will be generated.

1
Original areas will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old areas are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a scaled set of areas (and their corresponding keypoints, lines, and mesh) from a pattern of areas. The MAT, TYPE, REAL, and ESYS attributes are based on the areas in the pattern and not the current settings. Scaling is done in the active coordinate system. Areas in the pattern could have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Scale>Areas

ARSPLIT, *AREA*, *KP1*, *KP2*, *TOL*, *Factor*

Splits an area between two keypoints (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

AREA
The ID for the area to be split.

KP1
The ID of an existing keypoint within the area. This will be the starting keypoint for the boundary line. You cannot specify a keypoint for *KP1* that is already connected to *KP2*.

KP2
The ID of an existing keypoint within the area that will become the end point for the boundary line.

TOL
Label to activate user-definable area tolerance. If *TOL* = TIGHT user-definable tolerances are used and the tolerance factor is specified by *Factor*.

Factor
The user-definable tolerance factor. Valid entries are integer values from 1 (the tightest setting) to 10 (the loosest and default setting). This is only used when *TOL* = TIGHT.

Notes

The **ARSPLIT** command creates a new boundary line between the specified keypoints, splitting the specified area into two areas sharing that boundary. The command will allow you to specify a keypoint pair such that the

resultant boundary line will fall outside of the confines of the specified area; make sure that the keypoints you select do not create such a line.

Use this command to simplify the geometry of a model imported from a CAD file (this is a geometry "cleanup" tool). This tool is available only for models imported from CAD files (Default IGES option).

Normally, the default area tolerance is adequate for the **ARSPLIT** command. However, occasionally when attempting to split an extremely narrow area the **ARSPLIT** command can fail due to what the error message calls "numerical inadequacies" and you'll be prompted to try another location for the split. You may be able to split the area at the selected keypoints by tightening the area tolerance. To do this, issue the **ARSPLIT** command with *TOL* = TIGHT and *Factor* as an integer between 1 (the default, and loosest tolerance) and 10. You should remove any areas that are split through these arguments. If such areas remain, they may cause Boolean operations involving those areas to fail.

Menu Paths

Main Menu>Preprocessor>Modeling>Simplify>Toolkit>Split Areas

ARSYM, *Ncomp*, *NA1*, *NA2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Generates areas from an area pattern by symmetry reflection.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

Ncomp

Symmetry key:

X
: X symmetry (default).

Y
: Y symmetry.

Z
: Z symmetry.

NA1, *NA2*, *NINC*

Reflect areas from pattern beginning with *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is all selected areas [**ASEL**]. If *Ncomp* = P, use graphical picking to specify areas and ignore *NL2* and *NINC*. A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

KINC

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM

Specifies whether nodes and elements are also to be generated:

0
: Generate nodes and elements associated with the original areas, if they exist.

1
: Do not generate nodes and elements.

IMOVE
Specifies whether areas will be moved or newly defined:

0
Generate additional areas.

1
Move original areas to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old areas are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a reflected set of areas (and their corresponding keypoints, lines and mesh) from a given area pattern by a symmetry reflection (see analogous node symmetry command, **NSYM**). The MAT, TYPE, REAL, ESYS, and SECNUM attributes are based upon the areas in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. The active coordinate system must be a Cartesian system. Areas in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Areas are generated as described in the **AGEN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Areas

ASBA, *NA1*, *NA2*, *SEPO*, *KEEP1*, *KEEP2*

Subtracts areas from areas.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NA1
Area (or areas, if picking is used) to be subtracted from. If ALL, use all selected areas. Areas specified in this argument are not available for use in the *NA2* argument. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NA1*.

NA2
Area (or areas, if picking is used) to subtract. If ALL, use all selected areas (except those included in the *NA1* argument). A component name may also be substituted for *NA2*.

SEPO
Behavior if the intersection of the *NA1* areas and the *NA2* areas is a line or lines:

(blank)
The resulting areas will share line(s) where they touch.

SEPO
The resulting areas will have separate, but coincident line(s) where they touch.

KEEP1
Specifies whether *NA1* areas are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NA1* areas after **ASBA** operation (override **BOPTN** command settings).

KEEP
Keep *NA1* areas after **ASBA** operation (override **BOPTN** command settings).

KEEP2
Specifies whether *NA2* areas are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NA2* areas after **ASBA** operation (override **BOPTN** command settings).

KEEP
Keep *NA2* areas after **ASBA** operation (override **BOPTN** command settings).

Notes

Generates new areas by subtracting the regions common to both *NA1* and *NA2* areas (the intersection) from the *NA1* areas. The intersection can be an area(s) or line(s). If the intersection is a line and *SEP0* is blank, the *NA1* area is divided at the line and the resulting areas will be connected, sharing a common line where they touch. If *SEP0* is set to SEPO, *NA1* is divided into two unconnected areas with separate lines where they touch. See Solid Modeling in the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **ASBA**,ALL,ALL will have no effect since all the areas (in *NA1*) will be unavailable as *NA2* areas.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by Area
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by Area
Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Areas
Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>With Options>Areas

---

ASBL, *NA*, *NL*, *--*, *KEEPA*, *KEEPL*

Subtracts lines from areas.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NA
Area (or areas, if picking is used) to be subtracted from. If ALL, use all selected areas. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NA*.

NL
Line (or lines, if picking is used) to subtract. If ALL, use all selected lines. A component name may also be substituted for *NL*.

- -
Unused field.

KEEPA
Specifies whether *NA* areas are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NA* areas after **ASBL** operation (override **BOPTN** command settings).

KEEP
Keep *NA* areas after **ASBL** operation (override **BOPTN** command settings).

KEEPL
Specifies whether *NL* lines are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NL* lines after **ASBL** operation (override **BOPTN** command settings).

KEEP
Keep *NL* lines after **ASBL** operation (override **BOPTN** command settings).

Notes

Generates new areas by subtracting the regions common to both the areas and lines (the intersection) from the *NA* areas. The intersection will be a line(s). See Solid Modeling in the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by Line
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by Line

---

ASBV, *NA*, *NV*, *SEPO*, *KEEPA*, *KEEPV*

Subtracts volumes from areas.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NA
Area (or areas, if picking is used) to be subtracted from. If ALL, use all selected areas. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NA*.

NV
Volume (or volumes, if picking is used) to subtract. If ALL, use all selected volumes. A component name may also be substituted for *NV*.

SEP0
Behavior if the intersection of the areas and the volumes is a line or lines:

(blank)
The resulting areas will share line(s) where they touch.

SEPO
The resulting areas will have separate, but coincident line(s) where they touch.

KEEPA
Specifies whether *NA* areas are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NA* areas after **ASBV** operation (override **BOPTN** command settings).

KEEP
Keep *NA* areas after **ASBV** operation (override **BOPTN** command settings).

KEEPV
Specifies whether *NV* volumes are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete volumes after **ASBV** operation (override **BOPTN** command settings).

KEEP
Keep volumes after **ASBV** operation (override **BOPTN** command settings).

Notes

Generates new areas by subtracting the regions common to both *NA* areas and *NV* volumes (the intersection) from the *NA* areas. The intersection can be an area(s) or line(s). If the intersection is a line and *SEP0* is blank, the *NA* area is divided at the line and the resulting areas will be connected, sharing a common line where they touch. If *SEP0* is set to SEPO, *NA* is divided into two unconnected areas with separate lines where they touch. See Solid Modeling in the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by Volume
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by Volume
Main Menu>Preprocessor>Modeling>Operate>Divide>Area by Volu

ASBW, *NA*, *SEPO*, *KEEP*

Subtracts the intersection of the working plane from areas (divides areas).

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NA

Area (or areas, if picking is used) to be subtracted from. If *NA* = ALL, use all selected areas. If *NA* = P, graphical picking is enabled (valid only in the GUI). A component name may also be input for *NA*.

SEP0

Behavior of the created boundary.

(blank)

The resulting areas will share line(s) where they touch.

SEPO

The resulting areas will have separate, but coincident line(s).

KEEP

Specifies whether *NA* areas are to be deleted.

(blank)

Use the setting of KEEP on the **BOPTN** command.

DELETE

Delete *NA* areas after **ASBW** operation (override **BOPTN** command settings).

KEEP

Keep *NA* areas after **ASBW** operation (override **BOPTN** command settings).

Notes

Generates new areas by subtracting the intersection of the working plane from the *NA* areas. The intersection will be a line(s). The working plane must not be in the same plane as the *NA* areas(s). If *SEP0* is blank, the *NA* area is divided at the line and the resulting areas will be connected, sharing a common line where they touch. If *SEP0* is set to SEPO, *NA* is divided into two unconnected areas with separate lines. The SEPO option may cause unintended consequences if any keypoints exist along the cut plane. See Solid Modeling in the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Issuing the **ASBW** command under certain conditions may generate a topological degeneracy error. Do not issue the command if:

- A sphere or cylinder has been scaled. (A cylinder must be scaled unevenly in the XY plane.)
- A sphere or cylinder has not been scaled but the work plane has been rotated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by WrkPlane
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by WrkPlane
Main Menu>Preprocessor>Modeling>Operate>Divide>Area by WrkPlane

ASEL, *Type*, *Item*, *Comp*,*VMIN*, *VMAX*, *VINC*, *KSWP*

Selects a subset of areas.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type

Label identifying the type of select:

S
: Select a new set (default)

R
: Reselect a set from the current set.

A
: Additionally select a set and extend the current set.

U
: Unselect a set from the current set.

ALL
: Restore the full set.

NONE
: Unselect the full set.

INVE
: Invert the current set (selected becomes unselected and vice versa).

STAT
: Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Item

Label identifying data. Valid item labels are shown in ASEL - Valid Item and Component Labels. Some items also require a component label. If *Item* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to AREA.

Comp

Component of the item (if required). Valid component labels are shown in ASEL - Valid Item and Component Labels.

VMIN

Minimum value of item range. Ranges are area numbers, coordinate values, attribute numbers, etc., as appropriate for the item. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, or ESYS and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **ALIST** command for a discussion of signed attributes.

VMAX

Maximum value of item range. *VMAX* defaults to *VMIN*. If *VMAX* = *VMIN*, a tolerance of ±0.005 x *VMIN* is used, or ±1.0E-6 if *VMIN* = 0.0. If *VMAX* ≠ *VMIN*, a tolerance of ±1.0E-8 x (*VMAX*-*VMIN*) is used.

VINC

Value increment within range. Used only with integer ranges (such as for area numbers). Defaults to 1. *VINC* cannot be negative.

KSWP
Specifies whether only areas are to be selected:

0
Select areas only.

1
Select areas, as well as keypoints, lines, nodes, and elements associated with selected areas. Valid only with *Type* = S.

Command Default

All areas are selected.

Notes

Selects a subset of areas. For example, to select those areas with area numbers 1 through 7, use **ASEL**,S,AREA,,1,7. The selected subset is then used when the ALL label is entered (or implied) on other commands, such as **ALIST**,ALL. Only data identified by area number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

If *Item* = ACCA, the command selects only those areas that were created by concatenation. The *KSWP* field is processed, but the *Comp*, *VMIN*, *VMAX*, and *VINC* fields are ignored.

This command is valid in any processor.

ASEL - Valid Item and Component Labels

Valid Item and Component Labels ASEL, *Type Item,Comp,VMIN,VMAX,VINC,KSWP*

Item	Comp	Description
AREA		Area number.
EXT		Area numbers on exterior of selected volumes (ignore remaining fields).
LOC	X, Y, Z	X, Y, or Z center (picking "hot spot" location in the active coordinate system).
HPT		Area number (selects only areas with associated hard points).
MAT		Material number associated with the area.
TYPE		Element type number associated with the area.
REAL		Real constant set number associated with the area.
ESYS		Element coordinate system associated with the area.
SECN		Section number associated with the area.
ACCA		Concatenated areas (selects only areas that were created by area concatenation [**ACCAT**]).

Menu Paths

Utility Menu>Select>Entities

ASKIN, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Generates an area by "skinning" a surface through guiding lines.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NL1

The first guiding line forming the skinned area. If *NL1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NL1*. If *NL1* is negative, the line beginnings and ends will be used to direct the skinning of the remaining lines (see Changing the ASKIN Algorithm below).

NL2, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

The additional guiding lines for the skinned area (up to 9 total lines, including *NL1*, if using keyboard entry). If negative (and *NL1* is negative), the line beginning and end will be temporarily interchanged for the skinning operation (see Changing the ASKIN Algorithm below).

Notes

Generates an area by "skinning" a surface through specified guiding lines. The lines act as a set of "ribs" over which a surface is "stretched." Two opposite edges of the area are framed by the first (*NL1*) and last (*NLn*) guiding lines specified. The other two edges of the area are framed by splines-fit lines which the program automatically generates through the ends of all guiding lines. The interior of the area is shaped by the interior guiding lines. Once the area has been created, only the four edge lines will be attached to it. In rare cases, it may be necessary to change the default algorithm used by the **ASKIN** command (see Changing the ASKIN Algorithm below).

Changing the ASKIN Algorithm

When skinning from one guiding line to the next, the program can create the transition area in one of two ways: one more spiraled and one less spiraled ("flatter"). By default, the program attempts to produce the flatter transition, instead of the more spiraled transition. This algorithm can be changed by inputting *NL1* as a negative number, in which case the program connects all the keypoints at the line "beginnings" (**/PSYMB**,LDIR command) as one edge of the area, and all the line "ends" as the opposite edge, irrespective of the amount of spiraling produced in each transition area.

To further control the geometry of the area (if *NL1* is negative), the beginning and end of any specified line (other than *NL1*) can be temporarily interchanged (for the skinning operation only) by inputting that line number as negative. See Solid Modeling in the *ANSYS Modeling and Meshing Guide* for an illustration.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Skinning

---

ASLL, *Type*, *ARKEY*

Selects those areas containing the selected lines.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type

Label identifying the type of area select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

ARKEY
Specifies whether all contained area lines must be selected [**LSEL**]:

0
Select area if any of its lines are in the selected line set.

1
Select area only if all of its lines are in the selected line set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

ASLV, *Type*
Selects those areas contained in the selected volumes.

DATABASE: Selecting
MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of area select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

/ASSIGN, *Ident*, *Fname*, *Ext*, --

Reassigns a file name to an ANSYS file identifier.

SESSION: Files

MP ME ST DY <> PR EM <> FL PP ED

Ident

ANSYS file name identifier. Valid identifiers are: EMAT, ESAV, FULL, REDM, MODE, RDSP, RFRQ, TRI, RST, RTH, RMG, EROT, OSAV, RFL, and SELD. See the *ANSYS Basic Analysis Guide* for file descriptions. If blank, list currently reassigned files.

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Ext

Filename extension (8 character maximum).

--

Unused field

Notes

The reassignment of file names is valid only if it is done before the file is used. All file reassignments are retained (not cleared) even if the database is cleared [**/CLEAR**] or the Jobname is changed [**/FILNAME**]. Assigned files may be overwritten. If file name arguments (*Fname*, *Ext*, --) are blank, the default ANSYS assignment is restored. Use **SEOPT** for SUB files and **SEEXP** for DSUB files.

This command is valid only at the Begin Level.

Menu Paths

Utility Menu>File>ANSYS File Options

ASUB, *NA1*, *P1*, *P2*, *P3*, *P4*

Generates an area using the shape of an existing area.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NA1

Existing area number whose shape is to be used. If *P1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

P1

Keypoint defining starting corner of area.

P2
Keypoint defining second corner of area.

P3
Keypoint defining third corner of area.

P4
Keypoint defining fourth corner of area (defaults to *P3*).

Notes

The new area will overlay the old area. Often used when the area to be subdivided consists of a complex shape that was not generated in a single coordinate system. Keypoints and any corresponding lines must lie on the existing area. Missing lines are generated to lie on the given area. The active coordinate system is ignored.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>Overlaid on Area

---

ASUM, *LAB*

Calculates and prints geometry statistics of the selected areas.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

LAB
Controls the degree of tessellation used in the calculation of area properties. If *LAB* = DEFAULT, area calculations will use the degree of tessellation set through the **/FACET** command. If *LAB* = FINE, area calculations are based on a finer tessellation.

Notes

Calculates and prints geometry statistics (area, centroid location, moments of inertia, volume, etc.) associated with the selected areas. **ASUM** should only be used on perfectly flat areas.

Geometry items are reported in the global Cartesian coordinate system. A unit density (and thickness) is assumed unless the areas have a material (and real constant) association via the **AATT** command. The command always uses a unit density and a unit thickness for layered shell elements (such as SHELL91, SHELL99, SHELL131, SHELL163, and SHELL181).

Items calculated by **ASUM** and later retrieved via a ***GET** or ***VGET** command are valid only if the model is not modified after issuing the **ASUM** command.

Setting a finer degree of tessellation will provide area calculations with greater accuracy, especially for thin, hollow models. However, using a finer degree of tessellation requires longer processing.

For very narrow (sliver) areas, such that the ratio of the minimum to the maximum dimension is less than 0.01, the **ASUM** command can provide erroneous area information. To ensure that such calculations are accurate, make certain that you subdivide such areas so that the ratio of the minimum to the maximum is at least 0.05.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Areas

---

ATAN, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*

Forms the arctangent of a complex variable.

POST26: Operations

MP ME ST DY <> PR EM <> <> PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA
Reference number of the complex variable to be operated on.

--, --
Unused fields.

Name
Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --
Unused fields.

FACTA
Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0). Usually *FACTA* should be set to 1. *FACTA* may affect the position of the angle by a multiple of π, resulting in a quadrant change.

Notes

Forms the arctangent of a complex variable according to the operation:

$$IR = \text{ATAN}(FACTA \times b/a)$$

where *a* and *b* are the real and imaginary parts, respectively, of the complex variable *IA* (which is of the form $a + ib$). The arctangent represents the phase angle (in radians), and is valid only for a harmonic analysis (**ANTYPE**,HARMIC).

Since the scaling factor is applied uniformly to b/a, applying *any* positive or negative scaling factor will not affect the size of the phase angle, with the exception that a negative scaling factor will change the results quadrant by π. The magnitude of a complex number is still obtained through the **ABS** command. See the *ANSYS, Inc. Theory Reference* for details.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Arctangent

ATRAN, *KCNTO*, *NA1*, *NA2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Transfers a pattern of areas to another coordinate system.

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

KCNTO
Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system. The coordinate system type and parameters of *KCNTO* must be the same as the active system.

NA1, *NA2*, *NINC*
Transfer area pattern beginning with *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is all selected areas [**ASEL**]. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

KINC
Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM
Specifies whether elements and nodes are also to be generated:

0
Generate nodes and elements associated with the original areas, if they exist.

1
Do not generate nodes and elements.

IMOVE
Specifies whether to redefine the existing areas:

0
Generate additional areas.

1
Move original areas to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old areas are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Transfers a pattern of areas (and their corresponding lines, keypoints and mesh) from one coordinate system to another (see analogous node **TRANSFER** command). The MAT, TYPE, REAL, and ESYS attributes are based upon the areas in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Coordinate and slope values are interpreted in the active coordinate system and are transferred directly. Areas are generated as described in the **AGEN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Areas

ATYPE

Specifies "Analysis types" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Analysis Type

/AUTO, *WN*

Resets the focus and distance specifications to "automatically calculated."

GRAPHICS: Views

MP ME ST DY <> PR EM <> FL PP ED

WN

Window number (or ALL) to which command applies (defaults to 1).

Notes

Focus point and distance will be automatically calculated during next display. Settings may still be changed with the **/FOCUS** and **/DIST** commands after this command has been issued. See also the **/USER** command.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate
Utility Menu>PlotCtrls>View Settings>Automatic Fit Mode

AUTOTS, *Key*

Specifies whether to use automatic time stepping or load stepping.

SOLUTION: Load Step Options

MP ME ST <> <> PR EM <> <> PP ED

Key

Automatic time stepping key:

OFF
Do not use automatic time stepping.

ON
Use automatic time stepping.

Command Default

ANSYS determined time stepping when **SOLCONTROL**,ON. No automatic time stepping when **SOLCONTROL**,OFF.

Notes

Specifies whether to use automatic time stepping (or load stepping) over this load step. If *Key* = ON, both time step prediction and time step bisection will be used. Used only if *DTIME* (specified on the **DELTIM** command) is less than the time span or conversely, if *NSBSTP* (on the **NSUBST** command) is greater than one.

If you run an analysis with **SOLCONTROL**,ON, but do not issue the **AUTOTS** command, ANSYS will choose whether or not to use automatic time stepping. The program-chosen option will be recorded on the log file as **AUTOTS**,-1.

You cannot use automatic time stepping [**AUTOTS**], line search [**LNSRCH**], or the DOF solution predictor [**PRED**] with the arc-length method [**ARCLEN**, **ARCTRM**]. If you activate the arc-length method after you set **AUTOTS**, **LNSRCH**, or **PRED**, a warning message appears. If you choose to proceed with the arc-length method activation, ANSYS disables your automatic time stepping, line search, and DOF predictor settings.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substps
Main Menu>Solution>Analysis Type>Sol'n Controls>Basic
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substps

---

/AUX2

Enters the binary file dumping processor.

AUX2: Binary Files
SESSION: Processor Entry
MP ME ST DY <> PR EM <> FL PP ED

Notes

Enters the binary file dumping processor (ANSYS auxiliary processor AUX2). This processor is used to dump the contents of certain ANSYS binary files for visual examination.

This command is valid only at the Begin Level.

Menu Paths

Utility Menu>File>List>Binary Files
Utility Menu>List>Files>Binary Files

/AUX3

Enters the results file editing processor.

AUX3: Results Files
SESSION: Processor Entry

MP ME ST DY <> PR EM <> FL PP ED

Notes

Enters the results file editing processor (ANSYS auxiliary processor AUX3). This processor is used to edit ANSYS results files.

This command is valid only at the Begin Level.

Menu Paths

This command cannot be accessed from a menu.

/AUX12

Enters the radiation processor.

AUX12: Radiation Substructures
SESSION: Processor Entry

MP ME <> <> <> PR <> <> <> PP ED

Notes

Enters the radiation processor (ANSYS auxiliary processor AUX12). This processor supports the Radiation Matrix and the Radiosity Solver methods.

This command is valid only at the Begin Level.

Menu Paths

Main Menu>Radiation Matrix

/AUX15

Enters the IGES file transfer processor.

AUX15: IGES
POST1: Controls

MP ME ST DY <> PR EM <> FL PP ED

Notes

Enters the IGES file transfer processor (ANSYS auxiliary processor AUX15), used to read an IGES data file into the ANSYS program.

This command is valid only at the Begin Level.

Menu Paths

Utility Menu>File>Import

AVPRIN, *KEY*, *EFFNU*

Specifies how principal and vector sums are to be calculated.

POST1: Controls

MP ME ST DY <> PR EM <> <> PP ED

KEY

Averaging key:

0

Average the component values from the elements at a common node, then calculate the principal or vector sum from the averaged components (default).

1

Calculate the principal or vector sum values on a per element basis, then average these values from the elements at a common node.

EFFNU

Effective Poisson's ratio used for computing the von Mises equivalent strain (EQV). This command option is intended for use with line elements or in load case operations (**LCOPER**) only; ANSYS automatically selects the most appropriate effective Poisson's ratio, as discussed below.

Command Default

Average components at common node before principal or vector sum calculation.

Notes

Selects the method of combining components for certain derived nodal results when two or more elements connect to a common node. The methods apply to the calculations of derived nodal principal stresses, principal strains, and vector sums for selects, sorts, and output [**NSEL**, **NSORT**, **PRNSOL**, **PLNSOL**, etc.].

This command also defines the effective Poisson's ratio (*EFFNU*) used for equivalent strain calculations. If you use *EFFNU*, the default effective Poisson's ratios shown below will be overridden for all elements by the *EFFNU* value. To return to the default settings, issue the **RESET** command.

- From user input for EPEL and EPTH
- Set to 0.5 for EPPL and EPCR
- Set to 0.5 if the referenced material is hyperelastic
- Set to 0.0 for line elements (includes beam, link, and pipe elements, as well as discrete elements), cyclic symmetry analysis, and load case operations (**LCOPER**).

See the *ANSYS, Inc. Theory Reference* for more information.

Menu Paths

Main Menu>General Postproc>Element Table>Define Table
Main Menu>General Postproc>List Results>Nodal Solution
Main Menu>General Postproc>Options for Outp
Main Menu>General Postproc>Path Operations>Map onto Path
Main Menu>General Postproc>Path Operations>Map onto Path>FE Results
Main Menu>General Postproc>Plot Results>Contour Plot>Element Solu
Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu
Utility Menu>List>Results>Nodal Solution
Utility Menu>List>Results>Options
Utility Menu>Plot>Results>Contour Plot>Elem Solution
Utility Menu>Plot>Results>Contour Plot>Nodal Solution
Utility Menu>PlotCtrls>Multi-Plot Contrls

AVRES, *KEY*, *Opt*

Specifies how results data will be averaged when PowerGraphics is enabled.

POST1: Controls

MP ME ST DY <> PR EM <> FL PP ED

KEY

Averaging key.

1

Average results at all common subgrid locations.

2

Average results at all common subgrid locations except where material type [**MAT**] discontinuities exist. This option is the default.

3

Average results at all common subgrid locations except where real constant [**REAL**] discontinuities exist.

4

Average results at all common subgrid locations except where material type [**MAT**] or real constant [**REAL**] discontinuities exist.

Opt
Option to determine how results data are averaged.

(blank)
Average surface results data using only the exterior element faces (default).

FULL
Average surface results data using the exterior face and interior element data.

Notes

The **AVRES** command specifies how results data will be averaged at subgrid locations that are common to 2 or more elements. The command is valid only when PowerGraphics is enabled (via the **/GRAPHICS**,POWER command).

With PowerGraphics active (**/GRAPHICS**,POWER), the averaging scheme for surface data with interior element data included (**AVRES**,,FULL) and multiple facets per edge (**/EFACET**,2 or **/EFACET**,4) will yield differing minimum and maximum contour values depending on the Z-Buffering options (**/TYPE**,,6 or **/TYPE**,,7). When the Section data is not included in the averaging schemes (**/TYPE**,,7), the resulting absolute value for the midside node is significantly smaller.

PowerGraphics does not average your stresses across discontinuous surfaces. The normals for various planes and facets are compared to a tolerance to determine continuity. The *ANGLE* value you specify in the **/EDGE** command is the tolerance for classifying surfaces as continuous or "coplanar."

The command affects nodal solution contour plots (**PLNSOL**), nodal solution printout (**PRNSOL**), and subgrid solution results accessed through the Query Results function (under General Postprocessing) in the GUI.

The command has no effect on the nodal degree of freedom solution values (UX, UY, UZ, TEMP, etc.).

The command is also available in **/SOLU**.

Menu Paths

Main Menu>General Postproc>Options for Outp
Utility Menu>List>Results>Options

/AXLAB, *Axis*, *Lab*
Labels the X and Y axes on graph displays.

GRAPHICS: Graphs

MP ME ST DY <> PR EM <> FL PP ED

Axis
Axis specifier:

X
Apply label to X axis.

Y
Apply label to Y axis.

Lab
Axis label (user defined text up to 30 characters long). Leave blank to reestablish the default for *Axis* axis.

Command Default

Labels are determined by the program.

Notes

This command is valid in any processor.

Menu Paths

Main Menu>Drop Test>Time History>Graph Variables
Main Menu>Drop Test>Time History>List Variables
Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

B Commands

/BATCH, *Lab*

Sets the program mode to "batch."

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Lab

Specifies listing mode during a batch run:

LIST

The batch output will include a listing of the input file.

(blank)

Suppress input data listing.

Command Default

Interactive mode.

Notes

Sets the program mode to "batch" when included as the first line on an input file of ANSYS commands. For convenience, this command is automatically recorded on the log file (**Jobname.LOG**) at the beginning of an interactive ANSYS session so that the log file can be reused later for batch input.

Caution: This command should not be entered directly in an interactive ANSYS session since all subsequent keyboard input is simply copied to a file, without further processing or prompts from the program (use the "system break" to exit the ANSYS program if this occurs).

The recommended method for choosing batch mode, rather than using the **/BATCH** command, is to select the **Batch** simulation environment from the **Configure ANSYS Products** task in the ANSYS launcher, or the batch mode entry option on the ANSYS execution command when entering the program.

This command is valid only at the Begin Level.

Menu Paths

This command cannot be accessed from a menu.

BCSOPTION, --, *Memory_Option*, *Memory_Size*, --, --, *Solve_Info*

Sets memory option for the sparse solver.

SOLUTION: Analysis Options

MP ME ST DY <> PR EM <> FL PP ED

- -

Unused field

Memory_Option

DEFAULT
Use the default memory usage strategy for the sparse solver. For smaller jobs, the default strategy attempts to run in memory (incore) with no I/O. For larger jobs, the default strategy attempts to obtain enough memory for large jobs to run with optimal I/O performance. Optimal I/O for the sparse solver is obtained when sufficient workspace is allocated to keep the largest matrix front in memory during factorization. The amount of memory required for the sparse solver is unknown until the matrix structure is preprocessed, including equation reordering. The amount allocated for the sparse solver is then dynamically adjusted using the ANSYS memory manager.

INCORE
Use a memory allocation strategy in the sparse solver that will attempt to obtain enough memory to run with the entire factorized matrix in memory. If the sparse solver memory requirement exceeds 8 Gigabytes for real number matrices or 16 Gigabytes for complex number matrices, this option will revert to optimal I/O memory. This option is also subject to the memory size specified in *Memory_Size*.

Memory_Size
Sparse solver maximum memory size in Megabytes. If *Memory_Size* is not specified and *Memory_Option* = DEFAULT, the incore memory strategy is used for small problems. The optimal I/O mode is used for large problems. If *Memory_Size* is not specified and *Memory_Option* = INCORE, the program will attempt to use all available system memory to keep the factorized matrix in memory subject to an upper limit of one Gigabyte. To run a large analysis where sparse solver memory exceeds 1 Gigabyte, set *Memory_Size* to a larger value. The maximum value for *Memory_Size* for the sparse solver is 16 Gigabytes. *Memory_Size* is still subject to some adjustment if the size specified is too small or if the attempted memory allocation fails.

--
Unused field

--
Unused field

Solve_Info

OFF
Turns off additional output printing from the sparse solver (default).

PERFORMANCE
Turns on printing additional output from the sparse solver, including a performance summary (with a summary of file I/o for the sparse solver) and additional information on memory usage (for symbolic assembly).

Command Default

Automatic memory allocation is used.

Notes

This command provides the option to use either minimum memory (default) or maximum memory when the sparse solver is selected. Using maximum memory, if available, reduces I/O and shortens total solver elapsed time.

Running incore is best for jobs which comfortably fit within the limits of the physical memory on a given system.

If the sparse solver workspace exceeds physical memory size, the system will be forced into paging. In this case, optimal I/O (**BCSOPTION**,DEFAULT) memory would be more efficient.

Incore memory is best for jobs under 100,000 DOFs, especially analyses where repeated solves are performed for a single matrix factorization.

Users with very large memory systems may want to try incore runs for larger job to improve performance. Users with very small memory systems may need to run in DEFAULT memory mode, in which the program will most likely run with optimal I/O performance.

For repeated runs with the sparse solver, users may set the initial sparse solver memory allocation to the amount required for factorization. This strategy reduces the frequency of allocation and reallocation in the run to make the INCORE option fully effective. Users with very large memory systems may use the *Memory_Size* argument to increase the maximum size attempted for incore runs. The maximum workspace allowed for the sparse solver is eight Gigabytes for real matrix systems and 16 Gigabytes for complex systems.

Menu Paths

This command cannot be accessed from a menu.

BELLOW, *NLOC*, *LENG*, *STIFF*, *FLEX*, *ELEM*

Defines a bellows in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

NLOC
Node where bellows is to be placed. Defaults to current run starting point [**RUN**].

LENG
Length of bellows (defaults to average pipe OD).

STIFF
Axial stiffness value (defaults to that of equivalent straight pipe).

FLEX
Bending flexibility factor (defaults to 1.0).

ELEM
Element number to be assigned to bellows (defaults to the previous maximum element number (MAXEL) + 1).

Notes

Defines a bellows (straight pipe element (PIPE16) with adjusted specifications and loadings) at a given location in a piping run.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Define Pipes>Bellows

---

BEND, *NEL1*, *NEL2*, *RAD*, *NDIV*, *ESTRT*, *EINC*

Defines a bend in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

NEL1, *NEL2*
Element numbers of the two intersecting straight pipes. Defaults to the last two straight pipe elements nearest the intersection of the last two runs.

RAD
Bend radius. If LR, use long radius standard (1.5 x nominal diameter) (default). If SR, use short radius standard (1.0 x nominal diameter).

NDIV
Number of divisions (elements) along bend (defaults to 2). A node is generated at the end of each division.

ESTRT
Number to be assigned to first element of bend (defaults to MAXEL + 1).

EINC
Element number increment (defaults to 1).

Notes

Defines a bend of curved (elbow) pipe elements (PIPE18) in place of the intersection of two previously defined straight pipe elements [**RUN**]. Two new nodes are generated at the ends of the bend (at the tangency points). A node is also generated at the center of curvature point. The two straight pipes are automatically "shortened" to meet the ends of the bend. The bend specifications and loadings are taken from the corresponding two straight pipes. The flexibility factors are calculated from the internal pressure and EX (evaluated at TAVE) based on the current **PPRES** and **PTEMP** command specifications when the element is generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Define Pipes>Elbow

---

BETAD, *VALUE*

Defines the stiffness matrix multiplier for damping.

SOLUTION: Dynamic Options

MP ME ST <> <> <> <> <> <> PP ED

VALUE
Stiffness matrix multiplier for damping.

Notes

This command defines the stiffness matrix multiplier, β, in the damping expression given in the **ALPHAD** command. Values of β may also be input as a material property (use the DAMP label on the **MP** command). If DAMP is included, the DAMP value is added to the BETAD value as appropriate (see the *ANSYS, Inc. Theory Reference*). Damping is not used in the static (**ANTYPE**,STATIC) or buckling (**ANTYPE**,BUCKLE) analyses.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping
Main Menu>Solution>Analysis Type>Sol'n Controls>Transient
Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping

BF, *NODE*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *VAL4*
Defines a nodal body force load.

SOLUTION: FE Body Loads

MP ME ST <> <> PR EM <> FL PP ED

NODE
Node to which body load applies. If *NODE* = ALL, apply to all selected nodes [**NSEL**]. A component name may also be substituted for *NODE*.

Lab
Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. Structural labels: TEMP (temperature), FLUE (fluence). Thermal labels: HGEN (heat generation rate). Magnetic labels: TEMP (temperature), MVDI (magnetic virtual displacements flag). Electric labels: TEMP (temperature), CHRGD (charge density). High-frequency electromagnetic labels: JS (current density), H (magnetic field), EF (electric field), PORT (number 1–50 for interior waveguide or transmission line port). FLOTRAN labels: HGEN (heat generation rate), FORC (nodal body force densities in momentum equation).

VAL1, *VAL2*, *VAL3*
Value associated with the *Lab* item or table name reference for tabular boundary conditions. To specify a table, enclose the table name in percent signs (%) (e.g., **BF**,*NODE*,TEMP,%*tabname*%). Use the ***DIM** command to define a table. Use only *VAL1* for TEMP, FLUE, HGEN, MVDI, CHRGD, and PORT. If *Lab* = PORT, *VAL1* is a port number representing an interior waveguide or transmission line port. Use *VAL1*, *VAL2*, *VAL3* for the X, Y, and Z components of JS, H, EF, and FORC.

VAL4
If *Lab* = H or EF, *VAL4* is the phase angle in degrees. If *Lab* = JS, *VAL4* is the phase angle in degrees or a negative port number for a driven port.

Notes

Defines a nodal body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.). Nodal body loads default to the **BFUNIF** values, if they were previously specified.

You can specify a table name (*VAL* = %*tabname*%) only for temperature (TEMP), heat generation rate (HGEN), and nodal body force density (FORC) body load labels. When using TEMP, you can define a one-dimensional table that varies with respect to time (TIME) only. When defining this table, enter TIME as the primary variable. No other primary variables are valid.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppElecField>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppElecField>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Nodes
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Nodes
Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Nodes
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Nodes

---

BFA, *AREA*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *VAL4*
Defines a body force load on an area.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

AREA
Area to which body load applies. If ALL, apply to all selected areas [**ASEL**]. A component name may also be substituted for *AREA*.

Lab
Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. Structural labels: TEMP (temperature), FLUE (fluence). Thermal labels: HGEN (heat generation rate). Magnetic labels: TEMP (temperature), JS (current densities), VLTG (voltage drop). Electric labels: TEMP (temperature), CHRGD (charge density). High-frequency electromagnetic labels: JS (current density), H (magnetic field), EF (electric field), PORT (number 1–50 for interior waveguide or transmission line port). FLOTRAN label: HGEN (heat generation rate).

VAL1, *VAL2*, *VAL3*
Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, CHRGD, and PORT. If *Lab* = PORT, *VAL1* is a port number representing an interior waveguide or transmission line port. Use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components of JS, H, and EF. For *Lab* = VLTG, *VAL1* is the voltage drop and *VAL2* is the phase angle. When specifying a table name, you must enclose the table name in percent signs (%), e.g., **BFA**,*AREA*,*Lab*,%*tabname*%. Use the ***DIM** command to define a table.

VAL4

If *Lab* = H or EF, *VAL4* is the phase angle in degrees. If *Lab* = JS, *VAL4* is the phase angle in degrees or a negative port number for a driven port.

Notes

Defines a body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) on an area. Body loads may be transferred from areas to area elements (or to nodes if area elements do not exist) with the **BFTRAN** or **SBCTRAN** commands. Body loads default to the value specified on the **BFUNIF** command, if it was previously specified.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels. When using TEMP, you can define a one-dimensional table that varies with respect to time (TIME) only. When defining this table, enter TIME as the primary variable. No other primary variables are valid.

Body loads specified by the **BFA** command can conflict with other specified body loads. See Resolution of Conflicting Body Load Specifications in the *ANSYS Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppElecField>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppElecField>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Areas
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Areas
Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Areas
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Areas

BFADELE, *AREA*, *Lab*

Deletes body force loads on an area.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

AREA

Area at which body load is to be deleted. If ALL, delete for all selected areas [**ASEL**]. A component name may also be substituted for *AREA*.

Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. See the **BFA** command for labels.

Notes

Deletes body force loads (and all corresponding finite element loads) for a specified area and label. Body loads may be defined on an area with the **BFA** command.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelElecField>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelIntPort>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Areas
Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Areas
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Areas
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Areas
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelElecField>On Areas
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelIntPort>On Areas
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Areas
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Areas
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Areas
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Areas
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Areas
Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Areas
Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Areas
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Areas

BFALIST, *AREA*, *Lab*

Lists the body force loads on an area.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

AREA

Area at which body load is to be listed. If ALL (or blank), list for all selected areas [**ASEL**]. If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *AREA*.

Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. See the **BFA** command for labels.

Notes

Lists the body force loads for the specified area and label. Body loads may be defined on an area with the **BFA** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Body Loads>On All Areas
Utility Menu>List>Loads>Body Loads>On Picked Areas

BFCUM, *Lab*, *Oper*, *FACT*, *TBASE*

Specifies that nodal body force loads are to be accumulated.

SOLUTION: FE Body Loads

MP ME ST <> <> PR EM <> FL PP ED

Lab

Valid body load label. If ALL, use all appropriate labels. Structural labels: TEMP (temperature), FLUE (fluence). Thermal label: HGEN (heat generation rate). Magnetic labels: TEMP (temperature), JS (current densities), MVDI (magnetic virtual displacements flag). Electric labels: TEMP (temperature), CHRGD (charge density).

Oper

Accumulation key:

REPL
Subsequent values replace the previous values (default).

ADD
Subsequent values are added to the previous values.

IGNO
Subsequent values are ignored.

FACT

Scale factor for the nodal body load values. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor. The scale factor is not applied to body load phase angles.

TBASE

Used (only with *Lab* = TEMP) to calculate the temperature used in the add or replace operation (see *Oper*) as:

Temperature = *TBASE* + *FACT** (*T* - *TBASE*)

where *T* is the temperature specified on subsequent **BF** commands. *TBASE* defaults to zero.

Command Default

Replace previous values.

Notes

Allows repeated nodal body force loads to be replaced, added, or ignored. Nodal body loads are applied with the **BF** command. Issue the **BFLIST** command to list the nodal body loads. The operations occur when the next body loads are defined. For example, issuing the **BF** command with a temperature of 250 after a previous **BF** command with a temperature of 200 causes the new value of the temperature to be 450 with the add operation, 250 with the replace operation, or 200 with the ignore operation. A scale factor is also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a temperature of 700. The scale factor is applied even if no previous values exist. Issue **BFCUM**,STAT to show the current label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

Note — FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

BFCUM does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Nodal Body Ld
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Nodal Body Ld

---

BFDELE, *NODE*, *Lab*

Deletes nodal body force loads.

SOLUTION: FE Body Loads

MP ME ST DY <> PR EM <> FL PP ED

NODE

Node at which body load is to be deleted. If ALL, delete for all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab

Valid body load label. If ALL, use all appropriate labels. See the **BF** command for labels. In an explicit dynamic analysis, the only valid body load label is TEMP.

Notes

Deletes nodal body force loads for a specified node and label. Nodal body loads may be defined with the **BF** command (except in an explicit dynamic analysis).

The command **BFDELE**,TEMP can be used in an explicit dynamic analysis to delete temperature loads that are read in by the **LDREAD** command. **BFDELE** cannot be used to delete temperature loads defined by the **EDLOAD** command (use **EDLOAD**,DELE to delete this type of load).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelElecField>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelIntPort>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Body Forces
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Node Components
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Nodes
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Delete Temps
Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelElecField>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelIntPort>On Nodes
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Nodes
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Body Forces
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Nodes
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Nodes
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Nodes
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Node Components
Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Nodes
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Nodes
Main Menu>Solution>Loading Options>Delete Temps

BFE, *ELEM*, *Lab*, *STLOC*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

Defines an element body force load.

SOLUTION: FE Body Loads

MP ME ST <> <> PR EM <> FL PP ED

ELEM

Element to which body load applies. If ALL, apply to all selected elements [**ESEL**]. A component name may also be substituted for *ELEM*.

Lab

Valid body load label. Structural labels: TEMP (temperature), FLUE (fluence). Thermal labels: HGEN (heat generation rate). Magnetic labels: EF (electric field), TEMP (temperature), JS (current densities), VLTG (voltage drop), MVDI (magnetic virtual displacements flag). Electric labels: TEMP (temperature), CHRGD (charge density). High-frequency electromagnetic labels: JS (current density). Field volume interface label: FVIN (field volume interface flag). Valid labels are also listed for each element type in the *ANSYS Elements Reference* under "Body Loads" in the input table.

STLOC

Starting location for entering *VAL* data, below. For example, if *STLOC* = 1, data input in the *VAL1* field applies to the first element body load item available for the element type, *VAL2* applies to the second element item, etc. If *STLOC* = 5, data input in the *VAL1* field applies to the fifth element item, etc. Defaults to 1.

VAL1, *VAL2*, *VAL3*, *VAL4*

For *Lab* = TEMP, FLUE, HGEN, and CHRGD, *VAL1*--*VAL4* represent body load values at the starting location and subsequent locations (usually nodes) in the element. *VAL1* can also represent a table name for use with tabular boundary conditions. Enter only *VAL1* for a uniform body load across the element. For nonuniform loads, the values must be input in the same order as shown in the input table for the element type. Values initially default to the **BFUNIF** value (except for CHRGD which defaults to zero). For subsequent specifications, a blank leaves a previously specified value unchanged; if the value was not previously specified, the default value as described in the *ANSYS Elements Reference* is used.

For *Lab* = JS and *STLOC* = 1, *VAL1*, *VAL2* and *VAL3* are the X, Y, and Z components of current density (in the element coordinate system), and *VAL4* is the phase angle.

For *Lab* = EF and *STLOC* = 1, *VAL1*, *VAL2*, and *VAL3* are the X, Y, and Z components of electric field (in the global Cartesian coordinate system).

For *Lab* = VLTG and *STLOC* = 1, *VAL1* is the voltage drop and *VAL2* is the phase angle.

For *Lab* = FVIN, *VAL1* represents volume interface number for load transfer. *VAL2* -- *VAL4* are not valid.

When specifying a table, enclose the table name in percent signs (%), e.g., (**BFE**,*ELEM*,*Lab*, *Lab*,*STLOC*,%*tabname*%). Use the ***DIM** command to define a table.

Notes

Defines an element body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.). Body loads and element specific defaults are described for each element type in the *ANSYS Elements Reference*. If both the **BF** and **BFE** commands are used to apply a body load to an element, the **BFE** command takes precedence.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels. When using TEMP, you can define a one-dimensional table that varies with respect to time (TIME) only. When defining this table, enter TIME as the primary variable. No other primary variables are valid.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Volume Intr>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>Electric Field>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Elements
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Elements
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Elements
Main Menu>Solution>Define Loads>Apply>Field Volume Intr>On Elements
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Elements
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Elements
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Elements
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Elements
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Elements
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>Electric Field>On Elements
Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Elements
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Elements

---

BFECUM, *Lab*, *Oper*, *FACT*, *TBASE*

Specifies whether to ignore subsequent element body force loads.

SOLUTION: FE Body Loads

MP ME ST <> <> PR EM <> FL PP ED

Lab

Valid body load label. If ALL, use all appropriate labels. Structural labels: TEMP (temperature), FLUE (fluence). Thermal label: HGEN (heat generation rate). Magnetic labels: TEMP (temperature), JS (current densities), VLTG (voltage drop), MVDI (magnetic virtual displacements flag). Electric labels: TEMP (temperature), CHRGD (charge density).

Oper

Replace or ignore key:

REPL
Subsequent values replace the previous values (default).

IGNO
Subsequent values are ignored.

FACT
Scale factor for the element body load values. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor. The scale factor is not applied to body load phase angles.

TBASE
Used (only with *Lab* = TEMP) to calculate the temperature used in the add or replace operation (see *Oper*) as:

Temperature = *TBASE* + *FACT** (*T* - *TBASE*)

where *T* is the temperature specified on subsequent **BFE** commands. *TBASE* defaults to zero.

Command Default

Replace previous values.

Notes

Allows repeated element body force loads to be replaced or ignored. Element body loads are applied with the **BFE** command. Issue the **BFELIST** command to list the element body loads. The operations occur when the next body loads are defined. For example, issuing the **BFE** command with a temperature value of 25 after a previous **BFE** command with a temperature value of 20 causes the new value of that temperature to be 25 with the replace operation, or 20 with the ignore operation. A scale factor is also available to multiply the next value before the replace operation. A scale factor of 2.0 with the previous "replace" example results in a temperature of 50. The scale factor is applied even if no previous values exist. Issue **BFECUM**,STAT to show the current label, operation, and scale factors.

BFECUM does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Elem Body Lds
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Elem Body Lds

---

BFEDELE, *ELEM*, *Lab*

Deletes element body force loads.

SOLUTION: FE Body Loads

MP ME ST <> <> PR EM <> FL PP ED

ELEM
Element at which body load is to be deleted. If ALL, delete for all selected elements [A component name may also be substituted for *ELEM*.

Lab
Valid body load label. If ALL, use all appropriate labels. See **BFE** command for labels.

Notes

Deletes element body force loads for a specified element and label. Element body loads may be defined with the **BFE** commands.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Elems
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Element Components
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Elements
Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Elems
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Elements
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Elements
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Elements
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Elements
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Elements
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Elements
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Elements
Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Elements
Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Element Components
Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Elements
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Elements

BFELIST, *ELEM*, *Lab*

Lists the element body force loads.

SOLUTION: FE Body Loads

MP ME ST <> <> PR EM <> FL PP ED

ELEM

Element at which body load is to be listed. If ALL (or blank), list for all selected elements [**ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *ELEM*.

Lab

Valid body load label. If ALL (or blank), use all appropriate labels. See **BFE** command for labels.

Notes

Lists the element body force loads for the specified element and label. Element body loads may be defined with the **BFE** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Body Loads>On All Elements
Utility Menu>List>Loads>Body Loads>On Picked Elems

BFESCAL, *Lab*, *FACT*, *TBASE*

Scales element body force loads.

SOLUTION: FE Body Loads

MP ME ST <> <> PR EM <> FL PP ED

Lab

Valid body load label. If ALL, use all appropriate labels. Structural labels: TEMP (temperature), FLUE (fluence). Thermal label: HGEN (heat generation rate). Magnetic labels: TEMP (temperature), JS (current densities), VLTG (voltage drop), MVDI (magnetic virtual displacements flag). Electric labels: TEMP (temperature), CHRGD (charge density).

FACT

Scale factor for the element body load values. Zero (or blank) defaults to 1.0. Use a small number for a "zero" scale factor. The scale factor is not applied to body load phase angles.

TBASE

Base temperature for temperature difference. Used only with *Lab* = TEMP. Scale factor is applied to the temperature difference (*T* - *TBASE*) and then added to *TBASE*. *T* is the current temperature.

Notes

Scales element body force loads on the selected elements in the database. Issue the **BFELIST** command to list the element body loads. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled. (Note that such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.)

BFESCAL does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Elem Body Lds
Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Elem Body Lds

---

BFINT, *Fname1*, *Ext1*, *--*, *Fname2*, *Ext2*, *--*, *KPOS*, *Clab*, *KSHS*, *TOLOUT*, *TOLHGT*

Activates the body force interpolation operation.

POST1: Special Purpose

MP ME ST <> <> PR EM <> FL PP ED

Fname1
File name and directory path (248 characters maximum, including directory) from which to read data for interpolation. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext1
Filename extension (8 character maximum).

The extension defaults to NODE if *Fname1* is blank.

--
Unused field

Fname2
File name and directory path (248 characters maximum, including directory) to which **BF** commands are written. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext2
Filename extension (8 character maximum).

The extension defaults to BFIN if *Fname2* is blank.

--
Unused field

KPOS
Position on *Fname2* to write block of **BF** commands:

0
Beginning of file (overwrite existing file).

1
End of file (append to existing file).

Clab
Label (8 characters maximum, including the colon) for this block of **BF** commands in *Fname2*. This label is appended to the colon (:). Defaults to BF*n*, where *n* is the cumulative iteration number for the data set currently in the database.

KSHS
Shell-to-solid submodeling key:

0
Solid-to-solid or shell-to-shell submodel.

1
Shell-to-solid submodel.

TOLOUT
Extrapolation tolerance about elements, based on a fraction of the element dimension. Submodel nodes outside the element by more than *TOLOUT* are not accepted as candidates for DOF extrapolation. Defaults to 0.5 (50%).

TOLHGT
Height tolerance above or below shell elements, in units of length. Used only for shell-to-shell submodeling (*KSHS* = 0). Submodel nodes off of the element surface by more than *TOLHGT* are not accepted as candidates for DOF interpolation or extrapolation. Defaults to 0.0001 times the maximum element dimension.

Caution: Relaxing this tolerance to allow submodel nodes to be "found" could produce poor submodel results.

Notes

File *Fname1* should contain a node list for which body forces are to be interpolated [**NWRITE**]. File *Fname2* is created which contains interpolated body forces written as a block of nodal **BF** commands. Body forces are interpolated from elements having TEMP as a valid body force or degree of freedom, and only the label TEMP is written on the nodal **BF** commands. Interpolation is performed for all nodes on file *Fname1* using the results data currently in the database. The block of **BF** commands begins with an identifying colon label command and ends with a **/EOF** command. The colon label command is of the form :*Clab*, where *Clab* is described above. Interpolation from multiple results sets can be performed by looping through the results file in a user-defined macro. Additional blocks can be appended to *Fname2* by using *KPOS* and unique colon labels. A **/INPUT** command, with the appropriate colon label, may be used to read the block of commands.

Menu Paths

Main Menu>General Postproc>Submodeling>Interp Body Forc

BFK, *KPOI*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *PHASE*
Defines a body force load at a keypoint.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

KPOI
Keypoint to which body load applies. If ALL, apply to all selected keypoints [**KSEL**]. A component name may also be substituted for *KPOI*.

Lab
Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. Structural labels: TEMP (temperature), FLUE (fluence). Thermal labels: HGEN (heat generation rate). Magnetic labels: TEMP (temperature), JS (current densities), MVDI (magnetic virtual displacements flag). Electric labels: TEMP (temperature), CHRGD (charge density). High-frequency electromagnetic labels: JS (current density). All keypoints on a given area (or volume) must have the same **BFK** table name for the tables to be transferred to interior nodes.

VAL1, *VAL2*, *VAL3*
Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, MVDI and CHRGD. Use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components of JS. When specifying a table name, you must enclose the table name in percent signs (%), e.g., **BFK**,*KPOI*,*Lab*,%*tabname*%. Use the ***DIM** command to define a table.

PHASE
Phase angle in degrees associated with the JS label.

Notes

Defines a body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) at a keypoint. Body loads may be transferred from keypoints to nodes with the **BFTRAN** or **SBCTRAN** commands. Interpolation will be used to apply loads to the nodes on the lines between keypoints. All keypoints on a given area (or volume) must have the same **BFK** specification, with the same values, for the loads to be transferred to interior nodes in the area (or volume). If only one keypoint on a line has a **BFK** specification, the other keypoint defaults to the value specified on the **BFUNIF** command.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels. When using TEMP, you can define a one-dimensional table that varies with respect to time (TIME) only. When defining this table, enter TIME as the primary variable. No other primary variables are valid.

Body loads specified by the **BFK** command can conflict with other specified body loads. See Resolution of Conflicting Body Load Specifications in the *ANSYS Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Keypoints
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Keypoints
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Keypoints
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Keypoints

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Keypoints
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Keypoints
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Keypoints
Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Keypoints
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Keypoints

---

BFKDELE, *KPOI*, *Lab*

Deletes body force loads at a keypoint.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

KPOI
Keypoint at which body load is to be deleted. If ALL, delete for all selected keypoints [**KSEL**]. A component name may also be substituted for *KPOI*.

Lab
Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. See the **BFK** command for labels.

Notes

Deletes body force loads (and all corresponding finite element loads) for a specified keypoint and label. Body loads may be defined at a keypoint with the **BFK** command.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All KPs
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Keypoints
Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All KPs
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Keypoints
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Keypoints
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Keypoints
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Keypoints
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Keypoints
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Keypoints

Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Keypoints
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Keypoints

---

BFKLIST, *KPOI*, *Lab*

Lists the body force loads at keypoints.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

KPOI

Keypoint at which body load is to be listed. If ALL (or blank), list for all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*

Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. See the **BFK** command for labels.

Notes

Lists the body force loads for the specified keypoint and label. Keypoint body loads may be defined with the **BFK** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Body Loads>On All Keypoints
Utility Menu>List>Loads>Body Loads>On Picked KPs

BFL, *LINE*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

Defines a body force load on a line.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

LINE

Line to which body load applies. If ALL, apply to all selected lines [**LSEL**]. A component name may also be substituted for *LINE*.

Lab

Valid body load label. Load labels are listed under "Body loads" in the input table for each element type in the *ANSYS Elements Reference*. Structural labels: TEMP (temperature), FLUE (fluence). Thermal label: HGEN (heat generation rate). Magnetic labels: TEMP (temperature). Electric labels: TEMP (temperature), CHRGD (charge density). High-frequency electromagnetic labels: JS (current density), EF (electric field). FLOTRAN labels: HGEN (heat generation rate), FORC (body force density).

VAL1, *VAL2*, *VAL3*

Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, and CHRGD. Use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components of JS,

EF, and FORC. When specifying a table name, you must enclose the table name in percent signs (%), e.g., **BFL**,*LINE*,*Lab*,%*tabname*%. Use the ***DIM** command to define a table.

VAL4
If *Lab* = EF, *VAL4* is the phase angle in degrees. If *Lab* = JS, *VAL4* is the phase angle in degrees or a negative port number for a driven port.

Notes

Defines a body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) on a line. Body loads may be transferred from lines to line elements (or to nodes if line elements do not exist) with the **BFTRAN** or **SBCTRAN** commands.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels. When using TEMP, you can define a one-dimensional table that varies with respect to time (TIME) only. When defining this table, enter TIME as the primary variable. No other primary variables are valid.

Body loads specified by the **BFL** command can conflict with other specified body loads. See Resolution of Conflicting Body Load Specifications in the *ANSYS Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppElecField>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Lines
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Lines
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Lines
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppElecField>On Lines
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Lines
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Lines
Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Lines
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Lines

---

BFLDELE, *LINE*, *Lab*

Deletes body force loads on a line.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

LINE
Line at which body load is to be deleted. If ALL, delete for all selected lines [**LSEL**]. A component name may also be substituted for *LINE*.

Lab
Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. See the **BFL** command for labels.

Notes

Deletes body force loads (and all corresponding finite element loads) for a specified line and label. Body loads may be defined on a line with the **BFL** command.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelElecField>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Lines
Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Lines
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Lines
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Lines
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelElecField>On Lines
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Lines
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Lines
Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Lines
Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Lines
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Lines

---

BFLIST, *NODE*, *Lab*

Lists the body force loads on nodes.

SOLUTION: FE Body Loads

MP ME ST DY <> PR EM <> FL PP ED

NODE
Node at which body load is to be listed. If ALL (or blank), list for all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab
Valid body load label. If ALL (or blank), use all appropriate labels. See the **BF** command for labels. In an explicit dynamic analysis, the only valid body load label is TEMP.

Notes

Lists the body force loads for the specified node and label. Nodal body loads may be defined with the **BF** command (except in an explicit dynamic analysis).

The command **BFLIST**,TEMP can be used in an explicit dynamic analysis to list temperature loads that are read in by the **LDREAD** command. **BFLIST** cannot be used to list temperature loads defined by the **EDLOAD** command (use **EDLOAD**,LIST to list this type of load).

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Loading Options>List Temps
Main Menu>Solution>Loading Options>List Temps
Utility Menu>List>Loads>Body Loads>On All Nodes
Utility Menu>List>Loads>Body Loads>On Picked Nodes

BFLLIST, *LINE*, *Lab*

Lists the body force loads on a line.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

LINE
Line at which body load is to be listed. If ALL (or blank), list for all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*.

Lab
Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. See the **BFL** command for labels.

Notes

Lists the body force loads for the specified line and label. Body loads may be defined on a line with the **BFL** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Body Loads>On All Lines
Utility Menu>List>Loads>Body Loads>On Picked Lines

BFSCALE, *Lab*, *FACT*, *TBASE*

Scales body force loads at nodes.

SOLUTION: FE Body Loads

MP ME ST <> <> PR EM <> FL PP ED

Lab

Valid body load label. If ALL, use all appropriate labels. Structural labels: TEMP (temperature), FLUE (fluence). Thermal label: HGEN (heat generation rate). Magnetic labels: TEMP (temperature), JS (current densities), MVDI (magnetic virtual displacements flag). Electric label: TEMP (temperature), CHRGD (charge density).

FACT

Scale factor for the nodal body load values. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor. The scale factor is not applied to body load phase angles.

TBASE

Base temperature for temperature difference. Used only with *Lab* = TEMP. Scale factor is applied to the temperature difference (*T* - *TBASE*) and then added to *TBASE*. *T* is the current temperature.

Notes

Scales body force loads in the database on the selected nodes. Issue the **BFLIST** command to list the nodal body loads. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

Note — Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

BFSCALE does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Nodal Body Ld
Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Nodal Body Ld

---

BFTRAN

Transfers solid model body force loads to the finite element model.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

Notes

Body loads are transferred from selected keypoints and lines to selected nodes and from selected areas and volumes to selected elements. The **BFTRAN** operation is also done if the **SBCTRAN** command is either explicitly issued or automatically issued upon initiation of the solution calculations [**SOLVE**].

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Body Loads
Main Menu>Solution>Define Loads>Operate>Transfer to FE>Body Loads

---

BFUNIF, *Lab*, *VALUE*

Assigns a uniform body force load to all nodes.

SOLUTION: FE Body Loads

MP ME ST DY <> PR EM <> FL PP ED

Lab
Valid body load label. If ALL, use all appropriate labels. Structural labels: TEMP (temperature), FLUE (fluence). Explicit dynamics label: TEMP (temperature). Thermal labels: HGEN (heat generation rate), TEMP (initial temperatures).

VALUE
Uniform value associated with *Lab* item, or table name when specifying tabular boundary conditions. To specify a table, enclose the table name in percent signs (%), e.g., **BFUNIF**,*Lab*,%*tabname*%.

Command Default

Set TEMP to the reference temperature ([**TREF**] but not **MP**,REFT), and FLUE and HGEN to zero.

Notes

In a transient or nonlinear thermal analysis, the uniform temperature is used during the first iteration of a solution as follows: (a) as the starting nodal temperature (except where temperatures are explicitly specified [**D**, **DK**]), and (b) to evaluate temperature-dependent material properties. In a structural analysis or explicit dynamic analysis, the uniform temperature is used as the *default* temperature for thermal strain calculations and material property evaluation (except where body load temperatures are specified [**BF**, **BFE**, **BFK**, **LDREAD**]). In other scalar field analyses, the uniform temperature is used for material property evaluation.

When the command **BFUNIF**,TEMP is used in an explicit dynamic analysis, you cannot use the **EDLOAD**,TEMP command to apply temperature loading. Furthermore, any temperature loading defined by **BFUNIF** cannot be listed or deleted by the **EDLOAD** command.

An alternate command, **TUNIF**, may be used to set the uniform temperature instead of **BFUNIF**,TEMP.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels. When using TEMP, you can define a one-dimensional table that varies with respect to time (TIME) only. When defining this table, enter TIME as the primary variable. No other primary variables are valid. Tabular boundary conditions cannot be used in an explicit dynamic analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>Uniform Heat Gen
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>Uniform Fluen

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>Uniform Heat Gen
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>Uniform Heat Gen
Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>Uniform Fluen
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>Uniform Heat Gen

BFV, *VOLU*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *PHASE*

Defines a body force load on a volume.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

VOLU
Volume to which body load applies. If ALL, apply to all selected volumes [**VSEL**]. A component name may also be substituted for *VOLU*.

Lab
Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. Structural labels: TEMP (temperature), FLUE (fluence). Thermal labels: HGEN (heat generation rate). Magnetic labels: TEMP (temperature), JS (current densities), VLTG (voltage drop). Electric labels: TEMP (temperature), CHRGD (charge density). High-frequency electromagnetic labels: JS (current density). FLOTRAN label: HGEN (heat generation rate).

VAL1, *VAL2*, *VAL3*
Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, and CHRGD. Use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components of JS. For *Lab* = VLTG, *VAL1* is the voltage drop and *VAL2* is the phase angle. When specifying a table name, you must enclose the table name in percent signs (%), e.g., **BFV**,*VOLU*,*Lab*,%*tabname*%. Use the ***DIM** command to define a table.

PHASE
Phase angle in degrees associated with the JS label.

Notes

Defines a body force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) on a volume. Body loads may be transferred from volumes to volume elements (or to nodes if volume elements do not exist) with the **BFTRAN** or **SBCTRAN** commands. Body loads default to the value specified on the **BFUNIF** command, if it was previously specified.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels. When using TEMP, you can define a one-dimensional table that varies with respect to time (TIME) only. When defining this table, enter TIME as the primary variable. No other primary variables are valid.

Body loads specified by the **BFV** command can conflict with other specified body loads. See Resolution of Conflicting Body Load Specifications in the *ANSYS Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Fluence>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Volumes
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Volumes
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Volumes
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Volumes
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Volumes
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppVoltDrop>On Volumes
Main Menu>Solution>Define Loads>Apply>Structural>Other>Fluence>On Volumes
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Volumes

---

BFVDELE, *VOLU*, *Lab*

Deletes body force loads on a volume.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

VOLU
Volume at which body load is to be deleted. If ALL, delete for all selected volumes [**VSEL**]. A component name may also be substituted for *VOLU*.

Lab
Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. See the **BFV** command for labels.

Notes

Deletes body force loads (and all corresponding finite element loads) for a specified volume and label. Body loads may be defined on a volume with the **BFV** command.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Volms
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Volumes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Volumes
Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Volms
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Volumes
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Volumes
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Volumes
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>AppVoltDrop>On Volumes
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Volumes
Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Volumes
Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Volumes
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Volumes

---

BFVLIST, *VOLU*, *Lab*

Lists the body force loads on a volume.

SOLUTION: Solid Body Loads

MP ME ST <> <> PR EM <> <> PP ED

VOLU
Volume at which body load is to be listed. If ALL (or blank), list for all selected volumes [**VSEL**]. If *VOLU* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *VOLU*.

Lab
Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *ANSYS Elements Reference*. See the **BFV** command for labels.

Notes

Lists the body force loads for the specified volume and label. Body loads may be defined on a volume with the **BFV** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Body Loads>On All Volumes
Utility Menu>List>Loads>Body Loads>On Picked Volumes

BIOOPT

Specifies "Biot-Savart options" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Biot Savart Options

BIOT, *Label*

Calculates the Biot-Savart source magnetic field intensity.

SOLUTION: Misc Loads

MP ME ST <> <> <> EM <> <> PP ED

Label

Controls the Biot-Savart calculation:

NEW

Calculate the magnetic source field intensity (H_s) from the selected set of source elements to the selected set of nodes. Overwrite any existing H_s field values.

SUM

Calculate the H_s field from the selected set of source elements to the selected set of nodes. Accumulate with any existing H_s field values.

Command Default

Calculate the H_s field upon encountering the first **SOLVE** command to produce a source field.

Notes

Calculates the Biot-Savart source magnetic field intensity (H_s) at the selected nodes from the selected source elements. The calculation is done at the time the **BIOT** command is issued.

Source elements include primitives described by element SOURC36, and coupled-field elements SOLID5, LINK68, SOLID69, and SOLID98. Current conduction elements do not have a solved-for current distribution from which to calculate a source field until after the first substep. Inclusion of a current conduction element H_s field will require a subsequent **BIOT**,SUM command (with SOURC36 elements unselected) and a **SOLVE** command.

The units of H_s are as specified by the current **EMUNIT** command setting.

This command is also valid in PREP7.

The Biot-Savart source loading is handled automatically in the edge formulation; therefore, the **BIOT** command is ignored when using SOURC36 current loading with SOLID117 elements.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>Biot-Savart
Main Menu>Solution>Load Step Opts>Magnetics>Options Only>Biot-Savart

---

BLC4, *XCORNER*, *YCORNER*, *WIDTH*, *HEIGHT*, *DEPTH*

Creates a rectangular area or block volume by corner points.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

XCORNER, *YCORNER*
Working plane X and Y coordinates of one corner of the rectangle or block face.

WIDTH
The distance from *XCORNER* on or parallel to the working plane X-axis that, together with *YCORNER*, defines a second corner of the rectangle or block face.

HEIGHT
The distance from *YCORNER* on or parallel to the working plane Y-axis that, together with *XCORNER*, defines a third corner of the rectangle or block face.

DEPTH
The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the block. If *DEPTH* = 0 (default), a rectangular area is created on the working plane.

Notes

Defines a rectangular area anywhere on the working plane or a hexahedral volume with one face anywhere on the working plane. A rectangle will be defined with four keypoints and four lines. A volume will be defined with eight keypoints, twelve lines, and six areas, with the top and bottom faces parallel to the working plane. See the **BLC5**, **RECTNG**, and **BLOCK** commands for alternate ways to create rectangles and blocks.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Rectangle>By 2 Corners
Main Menu>Preprocessor>Modeling>Create>Volumes>Block>By 2 Corners & Z
Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Block>By 2 Corners & Z

BLC5, *XCENTER*, *YCENTER*, *WIDTH*, *HEIGHT*, *DEPTH*

Creates a rectangular area or block volume by center and corner points.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

XCENTER, *YCENTER*
Working plane X and Y coordinates of the center of the rectangle or block face.

WIDTH
The total distance on or parallel to the working plane X-axis defining the width of the rectangle or block face.

HEIGHT
The total distance on or parallel to the working plane Y-axis defining the height of the rectangle or block face.

DEPTH
The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the block. If *DEPTH* = 0 (default), a rectangular area is created on the working plane.

Note — If you are working with a model imported from an IGES file (import option set to DEFAULT), you must supply a value for *DEPTH* or the command is ignored.

Notes

Defines a rectangular area anywhere on the working plane or a hexahedral volume with one face anywhere on the working plane by specifying the center and corner points. A rectangle will be defined with four keypoints and four lines. A volume will be defined with eight keypoints, twelve lines, and six areas, with the top and bottom faces parallel to the working plane. See the **BLC4**, **RECTNG**, and **BLOCK** commands for alternate ways to create rectangles and blocks.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Rectangle>By Centr & Cornr
Main Menu>Preprocessor>Modeling>Create>Primitives>Block
Main Menu>Preprocessor>Modeling>Create>Volumes>Block>By Centr,Cornr,Z
Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Block>By Centr,Cornr,Z

BLOCK, *X1*, *X2*, *Y1*, *Y2*, *Z1*, *Z2*

Creates a block volume based on working plane coordinates.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

X1, *X2*
Working plane X coordinates of the block.

Y1, *Y2*
Working plane Y coordinates of the block.

Z1, *Z2*
Working plane Z coordinates of the block.

Notes

Defines a hexahedral volume based on the working plane. The block must have a spatial volume greater than zero (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) The volume will be defined with eight keypoints, twelve lines, and six areas, with the top and bottom faces parallel to the working plane. See the **BLC4** and **BLC5** commands for alternate ways to create blocks.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Block>By Dimensions
Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Block>By Dimensions

---

BOOL

Specifies "Booleans" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Solid Model

BOPTN, *Lab*, *Value*

Specifies Boolean operation options.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

Lab

Default/status key:

DEFA
: Resets settings to default values.

STAT
: Lists status of present settings.

Option to be controlled:

KEEP
: Delete or keep input entity option.

NUMB
Output numbering warning message option.

NWARN
No effect warning message option.

VERSION
Boolean compatibility option.

Value
Option settings if *Lab* = KEEP:

NO
Delete entities used as input with a Boolean operation (default). Entities will not be deleted if meshed or if attached to a higher entity.

YES
Keep input solid modeling entities.

Option settings if *Lab* = NUMB:

0
No warning message will be produced if the output entities of a Boolean operation are numbered based on geometry (default).

1
A warning message will be produced if the output entities of a Boolean operation are numbered based on geometry. (With geometric numbering, re-use of the input with altered dimensions (as in design optimization) may not produce the same numbering, and later operations in the input may fail or produce unexpected results.)

Option settings if *Lab* = NWARN:

0
A warning message will be produced if a Boolean operation has no effect (default).

1
No warning or error messages will be generated if a Boolean operation has no effect.

1
An error message will be produced if a Boolean operation has no effect.

Option settings if *Lab* = VERSION:

RV52
Activate the Revision 5.2 compatibility option (default). The 5.2 option can produce different numbering of the entities produced by Boolean operations than the 5.1 option. See Notes below.

RV51
Activate the Revision 5.1 compatibility option. The 5.1 option can produce different numbering of the entities produced by Boolean operations than the 5.2 option. See Notes below.

Command Default

Input entities will be deleted, and operations with no effect (i.e., operations which are valid but which do not cause a change in the input entities, such as adding two non-touching areas) will produce a warning message. The Revision 5.2 Boolean compatibility option will be used.

Notes

Boolean operations at Revision 5.2 may produce a different number of entities than previous revisions of ANSYS. When running input files created at earlier revisions of ANSYS, match the Boolean compatibility option (VERSION) to the revision originally used. For instance, if you are running Revision 5.2 and are reading an input file (**/INPUT**) created at Revision 5.1, it is recommended that you set VERSION to RV51 before reading the input.

See the *ANSYS Modeling and Meshing Guide* for further details on the functions of the RV51 and RV52 labels.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Settings

BRANCH, *NODE*, *X*, *Y*, *Z*

Defines the starting point for a piping branch.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

NODE

Start branch at this node.

X, *Y*, *Z*

Start branch at this location (in the active coordinate system). Used only if *NODE* is not input or input but the node itself is not previously defined. In either case a node is generated at this location and assigned the number *NODE* (or 1 + previous maximum node number if *NODE* is not input).

Notes

See the **RUN** command for information relating to piping models.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Start Branch>At Node
Main Menu>Preprocessor>Modeling>Create>Piping Models>Start Branch>At XYZ Loc

---

BSPLIN, *P1*, *P2*, *P3*, *P4*, *P5*, *P6*, *XV1*, *YV1*, *ZV1*, *XV6*, *YV6*, *ZV6*

Generates a single line from a spline fit to a series of keypoints.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

P1, *P2*, *P3*, *P4*, *P5*, *P6*

Keypoints through which a spline is fit. At least two keypoints must be defined. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

The following fields are used only if specified end slopes are desired; otherwise, zero curvature end slopes will be automatically calculated. The slope vector is parallel to a vector pointing from the origin of the active coordinate system [**CSYS**] to the position in space that XV, YV, ZV represents in that system.

XV1, *YV1*, *ZV1*
Orientation point of an outward vector tangent to line at *P1*. Vector coordinate system has its origin at the keypoint. Coordinate interpretation corresponds to the active coordinate system type, i.e., X is R for cylindrical, etc. Defaults to zero curvature slope.

XV6, *YV6*, *ZV6*
Orientation point of an outward vector tangent to a line at *P6* (or the last keypoint specified if fewer than six specified). Defaults to zero curvature slope.

Notes

One line is generated between keypoint *P1* and the last keypoint entered. The line will pass through each entered keypoint. Solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Splines>Spline thru KPs
Main Menu>Preprocessor>Modeling>Create>Lines>Splines>Spline thru Locs
Main Menu>Preprocessor>Modeling>Create>Lines>Splines>With Options>Spline thru KPs
Main Menu>Preprocessor>Modeling>Create>Lines>Splines>With Options>Spline thru Locs

---

BTOL, *PTOL*

Specifies the Boolean operation tolerances.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

PT0L
Point coincidence tolerance. Points within this distance to each other will be assumed to be coincident during Boolean operations. Loosening the tolerance will increase the run time and storage requirements, but will allow more Boolean intersections to succeed. Defaults to 0.10E-4.

Command Default

PT0L = 0.10E-4.

Notes

Use **BTOL**,DEFA to reset the setting to its default value. Use **BTOL**,STAT to list the status of the present setting.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Settings

BUCOPT, *Method*, *NMODE*, *SHIFT*, *LDMULTE*
Specifies buckling analysis options.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR <> <> <> PP ED

Method
Mode extraction method to be used (no default; you must specify a method):

SUBSP
Subspace iteration.

LANB
Block Lanczos.

NMODE
Number of modes to extract. (defaults to 1).

SHIFT
Shift point about which the eigenvalues are calculated (defaults to 0.0).

LDMULTE
Upper end of the load multiplier range of interest (defaults to 0).

Notes

This command is also valid in PREP7. If used in SOLUTION, this command is valid only within the first load step.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

C Commands

C*, *Comment*
Places a comment in the output.

SESSION: List Controls

MP ME ST DY <> PR EM <> FL PP ED

Comment
Comment string, up to 75 characters.

Notes

The output from this command consists of two lines -- a blank line followed by a line containing C*** and the comment. This command is similar to **/COM** except that the comment produced by **C*** is more easily identified in the output.

Another way to include a comment is to precede it with a ! character (on the same line). The ! may be placed anywhere on the line, and any input following it is ignored as a comment. No output is produced by such a comment, but the comment line is included on the log file. This is a convenient way to annotate the log file.

This command is valid anywhere.

Menu Paths

This command cannot be accessed from a menu.

CALC
Specifies "Calculation settings" as the subsequent status topic.

POST1: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Calculations

CBDOF, *Fname1*, *Ext1*, --, *Fname2*, *Ext2*, --, *KPOS*, *Clab*, *KSHS*, *TOLOUT*, *TOLHGT*

Activates cut boundary interpolation (for submodeling).

POST1: Special Purpose

MP ME ST <> <> PR EM <> FL PP ED

Fname1

File name and directory path (248 characters maximum, including directory) from which to read boundary node data. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext1

Filename extension (8 character maximum).

The extension defaults to NODE if *Fname1* is blank.

--

Unused field

Fname2

File name and directory path (248 characters maximum, including directory) to which cut boundary **D** commands are written. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext2

Filename extension (8 character maximum).

The extension defaults to CBDO if *Fname2* is blank.

--

Unused field

KPOS

Position on *Fname2* to write block of **D** commands:

0

Beginning of file (overwrite existing file).

1

End of file (append to existing file).

Clab

Label (8 characters maximum, including the colon) for this block of **D** commands on *Fname2*. This label is appended to the colon (:). Defaults to CB*n*, where *n* is the cumulative iteration number for the data set currently in the database. For imaginary data (see *KIMG* on the ***SET** command), *Clab* defaults to CI*n*.

KSHS

Shell-to-solid submodeling key:

0

Solid-to-solid or shell-to-shell submodel.

1

Shell-to-solid submodel.

TOLOUT
Extrapolation tolerance about elements, based on a fraction of the element dimension. Submodel nodes outside the element by more than *TOLOUT* are not accepted as candidates for DOF extrapolation. Defaults to 0.5 (50%).

TOLHGT
Height tolerance above or below shell elements, in units of length. Used only for shell-to-shell submodeling (*KSHS* = 0). Submodel nodes off of the element surface by more than *TOLHGT* are not accepted as candidates for DOF interpolation or extrapolation. Defaults to 0.0001 times the maximum element dimension.

Caution: Relaxing this tolerance to allow submodel nodes to be "found" could produce poor submodel results.

Notes

File *Fname1* should contain a node list for which boundary conditions are to be interpolated [**NWRITE**]. File *Fname2* is created which contains interpolated boundary conditions written as a block of **D** commands. Boundary conditions are written for the active degree of freedom set for the element from which interpolation is performed. Interpolation is performed on the selected set of elements. The block of **D** commands begins with an identifying colon label and ends with a **/EOF** command. The colon label is of the form :*Clab*, where *Clab* is described above. Interpolation from multiple results sets can be performed by looping through the results file in a user-defined macro. Additional blocks can be appended to *Fname2* by using *KPOS* and unique colon labels. A **/INPUT** command, with the appropriate colon label, may be used to read the block of commands.

Menu Paths

Main Menu>General Postproc>Submodeling>Interpolate DOF

CDOPT, *Option*

Specifies format to be used for archiving geometry.

PREP7: Database

MP ME ST DY <> PR EM <> FL PP ED

Option

IGES
Write solid model geometry information using IGES format (default).

ANF
Write solid model geometry information using ANSYS Neutral File format.

STAT
Print out the current format setting.

Notes

This command controls your solid model geometry format for **CDWRITE** operations. The ANF option affects only the COMB and SOLID options of the **CDWRITE** command. All other options remain unaffected.

This option setting is saved in the database.

Menu Paths

Main Menu>Preprocessor>Archive Model>Read
Main Menu>Preprocessor>Archive Model>Write

CDREAD, *Option*, *Fname*, *Ext*, --, *Fnamei*, *Exti*

Reads a file of solid model and database information into the database.

PREP7: Database

MP ME ST DY <> PR EM EH FL PP ED

Option
Selects which data to read:

ALL
Read all geometry, material property, load, and component data (default). Solid model geometry and loads will be read from the file ***Fnamei.Exti***. All other data will be read from the file ***Fname.Ext.***

DB
Read all database information contained in file ***Fname.Ext***. This file should contain all information mentioned above except the solid model loads. For geometry data, the element types must be defined, and they must use the same number of nodes.

SOLID
Read the solid model geometry and solid model loads from the file ***Fnamei.Exti***. This file could have been written by the **CDWRITE** or **IGESOUT** command.

COMB
Read the combined solid model and database information from the file ***Fname.Ext.***

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to CDB if *Fname* is blank.

--
Unused field

Fnamei
Name of the IGES file and its directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to *Fname*. Used only if *Option* = ALL or SOLID.

Exti
Filename extension (8 character maximum).

Defaults to IGES if *Fnamei* is blank.

Notes

This command causes coded files of solid model (in IGES format) and database (in command format) information to be read. These files are normally written by the **CDWRITE** or **IGESOUT** command. Note that the active coordinate system in these files has been reset to Cartesian (**CSYS**,0).

If a set of data exists prior to the **CDREAD** operation, that data set is offset upward to allow the new data to fit without overlap. The **NOOFFSET** command allows this offset to be ignored on a set-by-set basis, causing the existing data set to be overwritten with the new data set.

When you write the geometry data using the **CDWRITE**,GEOM option, you use the **CDREAD**,DB option to read the geometry information.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Archive Model>Read

CDWRITE, *Option*, *Fname*, *Ext*, --, *Fnamei*, *Exti*, *Fmat*

Writes geometry and load database items to a file.

PREP7: Database

MP ME ST DY <> PR EM <> FL PP ED

Option

Selects which data to write:

ALL

Write all appropriate geometry, material property, load, and component data (default). Two files will be produced. ***Fname.Ext*** will contain all data items mentioned in Notes, except the solid model data. ***Fnamei.Exti*** will contain the solid model geometry and solid model loads data in the form of IGES commands. This option is not valid when **CDOPT**, ANF is active.

COMB

Write all data mentioned, but to a single file, ***Fname.Ext***. Solid model geometry data will be written in either IGES or ANF format as specified in the **CDOPT**command, followed by the remainder of the data in the form of ANSYS commands. More information on these (IGES/ANF) file formats is provided below.

DB

Write all database information except the solid model and solid model loads to ***Fname.Ext*** in the form of ANSYS commands. This option is not valid when **CDOPT**, ANF is active.

SOLID

Write only the solid model geometry and solid model load data. This output will be in IGES or ANF format, as specified in the **CDOPT**command. More information on these (IGES/ANF) file formats is provided below.

GEOM

Write only element and nodal geometry data. No solid model geometry data will be written. One file, ***Fname.Ext***, will be produced. You use **CDREAD**,*DB* to read in a file written in this fashion.

CM

Write only node and element component and geometry data to ***Fname.Ext***.

MAT
Write only material property data (both linear and nonlinear) to ***Fname.Ext***.

LOAD
Write only loads for current load step to ***Fname.Ext***.

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Ext
Filename extension (8 character maximum).

The extension defaults to CDB if *Fname* is blank.

--
Unused field

Fnamei
Name of the IGES file and its directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to *Fname*. Used only if *Option* = ALL or SOLID. Previous data on this file, if any, is overwritten.

Exti
Filename extension (8 character maximum).

The extension defaults to IGES in all cases, except when **CDOPT**, ANF is active and your **CDWRITE**, *Option* = SOLID. In this case *Exti* = ANF.

Fmat
Format of the output file (defaults to BLOCKED).

BLOCKED
Blocked format. This format allows faster reading of the output file. The time savings is most significant when BLOCKED is used to read **.cdb** files associated with very large models.

UNBLOCKED
Unblocked format. Note that the **CDWRITE** output of components and assemblies will always be written in the BLOCKED style even if the UNBLOCKED option is specified.

Command Default

When **SOLCONTROL**,ON, does not write default values for most of the relevant solution control commands or options. When **SOLCONTROL**,OFF, writes all default values for solution control commands.

Default output format is determined by your **CDOPT** settings.

Notes

- IGES option to write solid model information:

  CDOPT, IGS (default)

Before you write your solid model entities, you must select all corresponding lower level entities (use **ALLSEL**, BELOW, ALL). You cannot access **CDWRITE** for models imported from IGES files using the FACETED translator (see the **IOPTN** command). If you issue **CDWRITE** after generating a beam mesh with orientation nodes, the database file will contain all of the nodes for every beam element, including the orientation nodes. However, the orientation keypoints that were specified for the line (**LATT**) are no longer associated with the line and won't be written out to the geometry file. All associativity between the line and the orientation keypoints is lost. The **CDWRITE** command with IGES format does not support (for beam meshing) any line operation that relies on solid model associativity. For example, meshing the areas adjacent to the meshed line, plotting the line that contains the orientation nodes, or clearing the mesh from the line that contains orientation nodes may not work as expected. Concatenated lines are not written. In IGES option, the line segments that make up the concatenated lines are written. However, if you encounter an area that contains a concatenated line, the write operation halts (that area can't be recreated during the read operation). If your model has areas that contain concatenated lines, you must first list these and then unconcatenate them before issuing the **CDWRITE** command. Similarly, hardpoint information cannot be written.

- ANF option to write solid model information:

  CDOPT, ANF

  All model information in the database (regardless of select status) is written to the archive file. However, when you restore the database using this archived file, the select status of entities will also be restored. This option restores all line attributes, including orientation keypoints. It also writes out any components (not assemblies) comprised of solid model entities. The ANF option halts **CDWRITE** when a concatenated line or an area that contains a concatenated line is detected. You must delete the concatenated lines before issuing **CDWRITE**. Similarly, hardpoint information cannot be written.

Load data includes the current load step only. Loads applied to the solid model (if any) are automatically transferred to the finite element model when this command is issued. **CDWRITE** writes out solid model loads for meshed models only. If the model is not meshed, ANSYS cannot save the solid model loads. Component data include component definitions, but not assembly definitions. Appropriate **NUMOFF** commands are included at the beginning of the file; this is to avoid overlap of an existing database when the file is read in.

Element order information (i.e., resulting from a **WAVES** command) is not written. The data in the database remain untouched.

The data may be reread (on a different machine, for example) with the **CDREAD** command. Caution: When the file is read in, the **NUMOFF**,MAT command may cause a mismatch between material definitions and material numbers referenced by certain loads and element real constants. See **NUMOFF** for details. Also, be aware that the files created by the **CDWRITE** command explicitly set the active coordinate system to Cartesian (**CSYS**,0).

You should generally use the blocked format (*Fmat* = BLOCKED) when writing out model data with **CDWRITE**. This is a compressed data format that greatly reduces the time required to read large models through the **CDREAD** command. The blocked and unblocked formats are described in Chapter 3 of the *Guide to Interfacing with ANSYS*.

If you use **CDWRITE** in any of the derived products (ANSYS Emag, ANSYS Professional), then before reading the file, you must edit the **Jobname.cdb** file to remove commands that are not available in the respective component product.

The **CDWRITE** command writes the PART information for any ANSYS LS-DYNA input file to the **Jobname.cdb** file using the **EDPREAD** command. Note that **EDPREAD** is not an ANSYS documented command, it is written only when the **CDWRITE** command is issued. The PART information can be automatically read into ANSYS with the **CDREAD** command. However, if more than one **Jobname.cdb** file is read, the PART list from the last **Job-**

name.cdb file overwrites the existing PART list of the total model. This will affect all PART-related commands contained in the **Jobname.cdb** file. That means the user can join models, but not PART-related inputs, which the user must modify using the newly-created PART numbers. In limited cases, an update of the PART list (**ED-WRITE**,PUPDATE) is possible. This requires that no used combination of MAT/TYPE/REAL appears more than once in the list. However, partial changes to the PART-related commands may be necessary.

Caution: The **CDWRITE** command does not support (for beam meshing) any line operation that relies on solid model associativity. For example, meshing the areas adjacent to the meshed line, plotting the line that contains the orientation nodes, or clearing the mesh from the line that contains orientation nodes may not work as expected. For more information about beam meshing, see Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide*.

This command is also valid in SOLUTION.

Product Restrictions

In ANSYS ED, *Option* = ALL, COMB, or SOLID is not allowed, and the default is *Option* = DB.

Menu Paths

Main Menu>Preprocessor>Archive Model>Write

CE, *NEQN*, *CONST*, *NODE1*, *Lab1*, *C1*, *NODE2*, *Lab2*, *C2*, *NODE3*, *Lab3*, *C3*

Defines a constraint equation relating degrees of freedom.

PREP7: Constraint Equations

MP ME ST DY <> PR EM <> <> PP ED

NEQN

Set equation reference number:

n
: Arbitrary set number.

HIGH
: The highest defined constraint equation number. This option is especially useful when adding nodes to an existing set.

NEXT
: The highest defined constraint equation number plus one. This option automatically numbers coupled sets so that existing sets are not modified.

The default value is HIGH.

CONST

Constant term of equation.

NODE1

Node for first term of equation. If -*NODE1*, this term is deleted from the equation.

Lab1

Degree of freedom label for first term of equation. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations, in radians). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Electric

labels: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials).

C1
Coefficient for first node term of equation. If zero, this term is ignored.

NODE2, *Lab2*, *C2*
Node, label, and coefficient for second term.

NODE3, *Lab3*, *C3*
Node, label, and coefficient for third term.

Notes

Repeat the **CE** command to add additional terms to the same equation. To change only the constant term, repeat the command with no node terms specified. Only the constant term can be changed during solution, and only with the **CECMOD** command.

Linear constraint equations may be used to relate the degrees of freedom of selected nodes in a more general manner than described for nodal coupling [**CP**]. The constraint equation is of the form:

$$\text{Constant} = \sum_{I=1}^{N} (\text{Coefficient(I)} * \text{U(I)})$$

where U(I) is the degree of freedom (displacement, temperature, etc.) of term (I). The following example is a set of two constraint equations, each containing three terms:

0.0 = 3.0* (1 UX) + 3.0* (4 UX) + (-2.0)* (4 ROTY)

2.0 = 6.0* (2 UX) + 10.0* (4 UY) + 1.0* (3 UZ)

The first unique degree of freedom in the equation is eliminated in terms of all other degrees of freedom in the equation. A unique degree of freedom is one which is not specified in any other constraint equation, coupled node set, specified displacement set, or master degree of freedom set. It is recommended that the first term of the equation be the degree of freedom to be eliminated. The first term of the equation cannot contain a master degree of freedom, and no term can contain coupled degrees of freedom. The same degree of freedom may be specified in more than one equation but care must be taken to avoid over-specification (over-constraint).

The degrees of freedom specified in the equation (i.e., UX, UY, ROTZ, etc.) must also be included in the model (as determined from the element types [**ET**]). Also, each node in the equation must be defined on an element (any element type containing that degree of freedom will do).

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Constraint Eqn

CECHECK, *ItemLab*, *Tolerance*, *DOF*

Check constraint equations and couplings for rigid body motions.

PREP7: Database
SOLUTION: Analysis Options

MP ME ST DY <> PR EM EH FL PP ED

ItemLab
Item indicating what is to be checked:

CE
Check constraint equations only

CP
Check couplings only

ALL
Check both CE and CP

Tolerance
Allowed amount of out-of-balance for any constraint equation or coupled set. The default value of 1.0e-6 is usually good.

DOF
Specifies which DOF is to be checked. Default is RIGID, the usual option. Other choices are individual DOF such as UX, ROTZ, etc. or THERM. The THERM option will check the constraint equations or coupled sets for free thermal expansions, whereas the individual DOFs check under rigid body motions. ALL is RIGID and THERM.

Notes

This command imposes a rigid body motion on the nodes attached to the constraint equation or coupled set and makes sure that no internal forces are generated for such rigid body motions. Generation of internal forces by rigid body motions usually indicates an error in the equation specification (possibly due to nodal coordinate rotations). The THERM option does a similar check to see that no internal forces are created by the equations if the body does a free thermal expansion (this check assumes a single isotropic coefficient of expansion).

Menu Paths

This command cannot be accessed from a menu.

CECMOD, *NEQN*, *CONST*

Modifies the constant term of a constraint equation during solution.

SOLUTION: Load Step Options

MP ME ST <> <> PR EM <> <> PP ED

NEQN
Reference number of constraint equation.

CONST
New value of the constant term of equation.

Notes

Other terms of the constraint equation cannot be changed during the solution phase, but must be defined or changed within PREP7 prior to the solution. See the **CE** command for details.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Modify ConstrEqn
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Modify ConstrEqn
Main Menu>Solution>Load Step Opts>Other>Modify ConstrEqn

CECYC, *Lowname*, *Highname*, *Nsector*, *HIndex*, *Tolerance*, *Kmove*, *Kpairs*

Generates the constraint equations for a cyclic symmetry analysis

PREP7: Constraint Equations

MP ME ST DY <> PR EM <> <> PP ED

Lowname
Name of a component for the nodes on the low angle edge of the sector. Enclosed in single quotes.

Highname
Name of a component for the nodes on the high angle edge of the sector. Enclosed in single quotes.

Nsector
Number of sectors in the complete 360 degrees.

HIndex
Harmonic index to be represented by this set of constraint equations. If *Hindex* is -1, generate constraint equations for static cyclic symmetry. If *HIndex* is -2, generate constraint equations for static cyclic asymmetry.

Tolerance
A positive tolerance is an absolute tolerance (length units), and a negative tolerance is a tolerance relative to the local element size.

Kmove

0
Nodes are not moved.

1
HIGHNAME component nodes are moved to match LOWNAME component nodes exactly.

Kpairs

0
Do not print paired nodes

1
Print table of paired nodes

Notes

The analysis can be either modal cyclic symmetry or static cyclic symmetry.

The pair of nodes for which constraint equations are written are rotated into **CSYS**,1.

Menu Paths

This command cannot be accessed from a menu.

CEDELE, *NEQN1*, *NEQN2*, *NINC*, *Nsel*

Deletes constraint equations.

PREP7: Constraint Equations

MP ME ST DY <> PR EM <> <> PP ED

NEQN1, *NEQN2*, *NINC*

Delete constraint equations from *NEQN1* to *NEQN2* (defaults to *NEQN1*) in steps of *NINC* (defaults to 1). If *NEQN1* = ALL, *NEQN2* and *NINC* will be ignored all constraint equations will be deleted.

Nsel

Additional node selection control:

ANY

Delete equation set if *any* of the selected nodes are in the set (default).

ALL

Delete equation set only if *all* of the selected nodes are in the set.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Del Constr Eqn

CEINTF, *TOLER*, *DOF1*, *DOF2*, *DOF3*, *DOF4*, *DOF5*, *DOF6*, *MoveTol*

Generates constraint equations at an interface.

PREP7: Constraint Equations

MP ME ST DY <> PR EM <> <> PP ED

TOLER

Tolerance about selected elements, based on a fraction of the element dimension (defaults to 0.25 (25%)). Nodes outside the element by more than the tolerance are not accepted as being on the interface.

DOF1, *DOF2*, *DOF3*, *DOF4*, *DOF5*, *DOF6*

Degrees of freedom for which constraint equations are written. Defaults to all applicable DOFs. *DOF1* accepts ALL as a valid label, in which case the rest are ignored (all DOFs are applied).

MoveTol

The allowed "motion" of a node (see Notes below). This distance is in terms of the element coordinates (-1.0 to 1.0). A typical value is 0.05. Defaults to 0 (do not move). *MoveTol* must be less than or equal to *TOLER*.

Notes

This command can be used to "tie" together two regions with dissimilar mesh patterns by generating constraint equations that connect the selected nodes of one region to the selected elements of the other region. At the interface between regions, nodes should be selected from the more dense mesh region, A, and the elements

selected from the less dense mesh region, B. The degrees of freedom of region A nodes are interpolated with the corresponding degrees of freedom of the nodes on the region B elements, using the shape functions of the region B elements. Constraint equations are then written that relate region A and B nodes at the interface.

The *MoveTol* field lets the nodes in the previously mentioned region A change coordinates when slightly inside or outside the elements of region B. The change in coordinates causes the nodes of region A to assume the same surface as the nodes associated with the elements of region B. The constraint equations that relate the nodes at both regions of the interface are then written.

Solid elements with six degrees of freedom should only be interfaced with other six degree-of-freedom elements. The region A nodes should be near the region B elements. A location tolerance based on the smallest region B element length may be input. Stresses across the interface are not necessarily continuous. Nodes in the interface region should not have specified constraints.

Use the **CPINTF** command to connect nodes by coupling instead of constraint equations. Use the **EINTF** command to connect nodes by line elements. See also the **NSEL** and **ESEL** commands for selecting nodes and elements. See the *ANSYS, Inc. Theory Reference* for a description of 3-D space used to determine if a node will be considered by this command.

As an alternative to the **CEINTF** command, you can use contact elements and the internal multipoint constraint (MPC) algorithm to tie together two regions having disimilar meshes. See Solid-solid and Shell-shell Assemblies for more information.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Adjacent Regions

CELIST, *NEQN1*, *NEQN2*, *NINC*, *Nsel*

Lists the constraint equations.

PREP7: Constraint Equations

MP ME ST DY <> PR EM <> <> PP ED

NEQN1, *NEQN2*, *NINC*

List constraint equations from *NEQN1* to *NEQN2* (defaults to *NEQN1*) in steps of *NINC* (defaults to 1). If *NEQN1* = ALL (default), *NEQN2* and *NINC* are ignored and all constraint equations are listed.

Nsel

Node selection control:

ANY

List equation set if *any* of the selected nodes are in the set (default).

ALL

List equation set only if *all* of the selected nodes are in the set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Other>Constraint Eqns>All CE nodes selected
Utility Menu>List>Other>Constraint Eqns>Any CE node selected

CENTER, *NODE*, *NODE1*, *NODE2*, *NODE3*, *RADIUS*

Defines a node at the center of curvature of 2 or 3 nodes.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

NODE
Number to be assigned to the node generated at the center of curvature.

NODE1, *NODE2*, *NODE3*
Three nodes used to calculated the center of curvature, as described under *RADIUS*.

RADIUS
Used to control the interpretation of *NODE1*, *NODE2* and *NODE3*:

0
NODE1, *NODE2* and *NODE3* lie on a circular arc. The program will calculate the center of curvature (and radius) (default).

$\neq 0$
NODE1 and *NODE2* are the endpoints of an arc, and *RADIUS* is the radius of curvature. The program will locate the center of curvature on the *NODE3* side of the *NODE1*-*NODE2* line if *RADIUS* > 0, and opposite to *NODE3* if *RADIUS* < 0.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>At Curvature Ctr

---

CEQN

Specifies "Constraint equations" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Constraint Eqns

CERIG, *MASTE*, *SLAVE*, *Ldof*, *Ldof2*, *Ldof3*, *Ldof4*, *Ldof5*

Defines a rigid region.

PREP7: Constraint Equations

MP ME ST DY <> PR <> <> <> PP ED

MASTE
Retained (or master) node for this rigid region. If *MASTE* = P, then graphical picking of the master and slave nodes is enabled (first node picked will be the master node, and subsequent nodes picked will be slave nodes), and subsequent fields are ignored (valid only in GUI).

SLAVE
Removed (or slave) node for this rigid region. If ALL, slave nodes are all selected nodes.

Ldof
Degrees of freedom associated with equations:

ALL
All applicable degrees of freedom (default). If 3-D, generate 6 equations based on UX, UY, UZ, ROTX, ROTY, ROTZ; if 2-D, generate 3 equations based on UX, UY, ROTZ.

UXYZ
Translational degrees of freedom. If 3-D, generate 3 equations based on the slave nodes' UX, UY, and UZ DOFs and the master node's UX, UY, UZ, ROTX, ROTY, and ROTZ DOFs; if 2-D, generate 2 equations based on the slave nodes UX and UY DOFs and the master nodes UX, UY, and ROTZ DOFs. No equations are generated for the rotational coupling.

RXYZ
Rotational degrees of freedom. If 3-D, generate 3 equations based on ROTX, ROTY, ROTZ; if 2-D, generate 1 equation based on ROTZ. No equations are generated for the translational coupling.

UX
Slave translational UX degree of freedom only.

UY
Slave translational UY degree of freedom only.

UZ
Slave translational UZ degree of freedom only.

ROTX
Slave rotational ROTX degree of freedom only.

ROTY
Slave rotational ROTY degree of freedom only.

ROTZ
Slave rotational ROTZ degree of freedom only.

Ldof2, *Ldof3*, *Ldof4*, *Ldof5*
Additional degrees of freedom. Used only if more than one degree of freedom required and *Ldof* is not ALL, UXYZ, or RXYZ.

Notes

Defines a rigid region (link, area or volume) by automatically generating constraint equations to relate nodes in the region. Nodes in the rigid region must be assigned a geometric location before this command is used. Also, nodes must be connected to elements having the required degree of freedom set (see *Ldof* above). Generated

constraint equations are based on small deflection theory. Generated constraint equations are numbered beginning from the highest previously defined equation number (*NEQN*) plus 1. Equations, once generated, may be listed [**CELIST**] or modified [**CE**] as desired. Repeat **CERIG** command for additional rigid region equations.

This command will generate the constraint equations needed for defining rigid lines in 2-D or 3-D space. Multiple rigid lines relative to a common point are used to define a rigid area or a rigid volume. In 2-D space, with *Ldof* = ALL, three equations are generated for each pair of constrained nodes. These equations define the three rigid body motions in global Cartesian space, i.e., two in-plane translations and one in-plane rotation. These equations assume the X-Y plane to be the active plane with UX, UY, and ROTZ degrees of freedom available at each node. Other types of equations can be generated with the appropriate *Ldof* labels.

Six equations are generated for each pair of constrained nodes in 3-D space (with *Ldof* = ALL). These equations define the six rigid body motions in global Cartesian space. These equations assume that UX, UY, UZ, ROTX, ROTY, and ROTZ degrees of freedom are available at each node.

The UXYZ label allows generating a partial set of rigid region equations. This option is useful for transmitting the bending moment between elements having different degrees of freedom at a node. With this option only two of the three equations are generated for each pair of constrained nodes in 2-D space. In 3-D space, only three of the six equations are generated. In each case the rotational coupling equations are not generated. Similarly, the RXYZ label allows generating a partial set of equations with the translational coupling equations omitted.

Applying this command to a large number of slave nodes may result in constraint equations with a large number of coefficients. This may significantly increase the peak memory required during the process of element assembly. If real memory or virtual memory is not available, consider reducing the number of slave nodes.

As an alternative to the **CERIG** command, you can define a similar type of rigid region using contact elements and the internal multipoint constraint (MPC) algorithm. See Surface-based Constraints for more information.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Rigid Region

CESGEN, *ITIME*, *INC*, *NSET1*, *NSET2*, *NINC*

Generates a set of constraint equations from existing sets.

PREP7: Constraint Equations

MP ME ST DY <> PR EM <> FL PP ED

ITIME, *INC*

Do this generation operation a total of *ITIME*s, incrementing all nodes in the existing sets by *INC* each time after the first. *ITIME* must be >1 for generation to occur.

NSET1, *NSET2*, *NINC*

Generate sets from sets beginning with *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* is negative, *NSET2* and *NINC* are ignored and the last |*NSET1*| sets (in sequence from maximum set number) are used as the sets to be repeated.

Notes

Generates additional sets of constraint equations (with same labels) from existing sets. Node numbers between sets may be uniformly incremented.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Gen w/Same DOF

CFACT, *RFACTA*, *IFACTA*, *RFACTB*, *IFACTB*, *RFACTC*, *IFACTC*
Defines complex scaling factors to be used with operations.

POST26: Controls

MP ME ST DY <> PR EM <> <> PP ED

RFACTA
Real portion of the complex scale factor used in place of *FACTA*.

IFACTA
Imaginary portion of the complex scale factor used in place of *FACTA*.

RFACTB
Real portion of the complex scale factor used in place of *FACTB*.

IFACTB
Imaginary portion of the complex scale factor used in place of *FACTB*.

RFACTC
Real portion of the complex scale factor used in place of *FACTC*.

IFACTC
Imaginary portion of the complex scale factor used in place of *FACTC*.

Command Default

Use the real factors as described with the operation command.

Notes

Defines complex scale factors to be used with the operations [**ADD**, **PROD**, etc.]. If this command is supplied, these complex factors override any real factors (*FACTA*, *FACTB*, *FACTC*) supplied on the operation commands. Factors are typically involved in scaling a specified variable, such as in the term *FACTA* x *IA* of the **ADD** command to scale variable *IA* before the ADD operation.

When the **CFACT** command is active, defaults are as follows: 1) if the complex factor is not specified, but the variable upon which it acts (such as *IA*) is specified, the factor defaults to 1.0+*i*0.0; 2) if the variable upon which the factor operates is not specified, but the factor is specified, the variable defaults to 1.0 so that the term in the operation becomes the complex factor itself; 3) if neither the factor nor the variable number is supplied, the term is omitted from the operation. Once the operation (such as the **ADD** command) has been processed, the **CFACT** command becomes inactive and must be specified again if it is to be used.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Complx ScaleFact

/CFORMAT, *NFIRST*, *NLAST*

Controls the graphical display of alphanumeric character strings for parameters, components, assemblies, and tables.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

NFIRST
Display the first *n* characters of the parameter, component, assembly, or table name, up to 32. Defaults to 32.

NLAST
Display the last *n* characters of the parameter, component, assembly, or table name, up to 32. Defaults to 0.

Notes

Use this command to control the length of the character string that is shown in the graphics window for a parameter, component, assembly, or table name.

The total number of characters (*NFIRST* + *NLAST* +3) cannot exceed 32.

If *NFIRST* is greater than zero and *NLAST* = 0, only the *NFIRST* characters are displayed, followed by an ellipsis.

If *NFIRST*= 0 and *NLAST* is greater than zero, only the *NLAST* characters are displayed, preceded by an ellipsis (...).

If both *NFIRST* and *NLAST* are greater than zero , the name will be shown as *NFIRST*, followed by an ellipsis (...), followed by *NLAST*, up to a maximum of 32 characters.

For example, if *NFIRST* = 6 and *NLAST* = 3, and the character string is LENGTHOFSIDEONE, then it will appear in the graphics window as LENGTH...ONE.

If the actual length of the character string is less than the specified combination of *NFIRST* + *NLAST* +3, then the actual string will be used.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

CGLOC, *XLOC*, *YLOC*, *ZLOC*

Specifies the origin location of the acceleration coordinate system.

SOLUTION: Inertia

MP ME ST DY <> PR <> <> FL PP ED

XLOC, *YLOC*, *ZLOC*
Global Cartesian X, Y, and Z coordinates of the acceleration coordinate system origin.

Notes

Specifies the origin location of the acceleration coordinate system with respect to the global Cartesian system. The axes of this acceleration coordinate system are parallel to the global Cartesian axes.

A structure may be rotating about the global Cartesian origin [**OMEGA**, **DOMEGA**], which may in turn be rotating about another point (the origin of the acceleration coordinate system), introducing Coriolis effects. The location of this point (relative to the global Cartesian origin) is specified with this **CGLOC** command. For example, if Y is vertical and the global system origin is at the surface of the earth while the acceleration system origin is at the center of the earth, *YLOC* should be -4000 miles (or equivalent) if the rotational effects of the earth are to be included. The rotational velocity of the global Cartesian system about this point is specified with the **CGOMGA** command, and the rotational acceleration is specified with the **DCGOMG** command.

The rotational velocities and accelerations are mainly intended to include mass effects in a static (**ANTYPE**,STATIC) analysis. If used in dynamic analyses, no coupling exists between the user input terms and the time history response of the structure. See the *ANSYS, Inc. Theory Reference* for details. Related commands are **ACEL**, **CGOMGA**, **DCGOMG**, **DOMEGA**, and **OMEGA**.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Rotating Coords
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Coriolis Effects
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Coriolis Effects
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Acceleration CS>Delete Accel CS
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Acceleration CS>Set Accel CS
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Coriolis Effects
Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Coriolis Effects
Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Rotating Coords
Main Menu>Solution>Loading Options>Acceleration CS>Delete Accel CS
Main Menu>Solution>Loading Options>Acceleration CS>Set Accel CS

CGOMGA, *CGOMX*, *CGOMY*, *CGOMZ*

Specifies the rotational velocity of the global origin.

SOLUTION: Inertia

MP ME ST <> <> PR <> <> FL PP ED

CGOMX, *CGOMY*, *CGOMZ*
Rotational velocity of the global origin about the acceleration system X, Y, and Z axes.

Notes

Specifies the rotational velocity of the global origin about each of the acceleration coordinate system axes. The location of the acceleration coordinate system is defined with the **CGLOC** command. Rotational velocities may be defined in analysis types **ANTYPE**,STATIC, HARMIC (full or mode superposition), TRANS (full or mode superposition), and SUBSTR. See the *ANSYS, Inc. Theory Reference* for details. Units are radians/time. Related commands are **ACEL**, **CGLOC**, **DCGOMG**, **DOMEGA**, and **OMEGA**.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Rotating Coords
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Coriolis Effects
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Coriolis Effects
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Coriolis Effects
Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Coriolis Effects
Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Rotating Coords

---

CHECK, *Sele*, *Levl*

Checks current database items for completeness.

PREP7: Database
SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> FL PP ED

Sele
Specifies which elements are to be checked:

(blank)
Check all data.

ESEL
Check only elements in the selected set and unselect any elements not producing geometry check messages. The remaining elements (those producing check messages) can then be displayed and corrected. A null set results if no elements produce a message. Issue **ESEL**,ALL to select all elements before proceeding.

Levl
Used only with *Sele* = ESEL:

WARN
Select elements producing warning and error messages.

ERR
Select only elements producing error messages (default).

Notes

This command will not work if **SHPP**,OFF has been set. A similar, automatic check of all data is done before the solution begins.

If the "Check Elements" option is invoked through the GUI (menu path **Main Menu> Preprocessor> Meshing> Check Elems**), the **CHECK**,ESEL logic is used to highlight elements in the following way: good elements are blue, elements having warnings are yellow, and bad (error) elements are red.

Note — The currently selected set of elements is not changed by this GUI function.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Meshing>Check Mesh>Individual Elm>Select Warning/Error Elements

---

CHKMSH, *Comp*

Checks area and volume entities for previous meshes.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

Comp
Name of component containing areas or volumes.

Notes

CHKMSH invokes a predefined ANSYS macro that checks areas and volumes to find out if they were previously meshed. This macro name will appear in the log file (**Jobname.LOG**) prior to area and volume meshing operations initiated through the GUI. This command is not intended to be typed in directly in an ANSYS session (although it can be included in an input file for use with the **/INPUT** command).

Menu Paths

This command cannot be accessed from a menu.

CIRCLE, *PCENT*, *RAD*, *PAXIS*, *PZERO*, *ARC*, *NSEG*

Generates circular arc lines.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

PCENT
Keypoint defining the center of the circle (in the plane of the circle). If *PCENT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

RAD
Radius of the circle. If *RAD* is blank and *PCENT* = P, the radius is the distance from *PCENT* to *PZERO*.

PAXIS
Keypoint defining axis of circle (along with *PCENT*). If *PCENT* = P and *PAXIS* is omitted, the axis is normal to the working plane.

PZERO
Keypoint defining the plane normal to circle (along with *PCENT* and *PAXIS*) and the zero degree location. Need not be in the plane of the circle. This value is not required if *PAXIS* is defined along the Y axis (that is, a circle in the XZ plane).

ARC
Arc length (in degrees). Positive follows right-hand rule about *PCENT-PAXIS* vector. Defaults to 360°.

NSEG
Number of lines around circumference (defaults to minimum required for 90°-maximum arcs, i.e., 4 for 360°). Number of keypoints generated is *NSEG* for 360° or *NSEG*+1 for less than 360°.

Notes

Generates circular arc lines (and their corresponding keypoints). Keypoints are generated at regular angular locations (based on a maximum spacing of 90°). Arc lines are generated connecting the keypoints. Keypoint and line numbers are automatically assigned, beginning with the lowest available values [**NUMSTR**]. Adjacent lines use a common keypoint. Line shapes are generated as arcs, regardless of the active coordinate system. Line shapes are invariant with coordinate system after they are generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>By Cent & Radius
Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>Full Circle

---

/CLABEL, *WN*, *KEY*

Specifies contour labeling.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

KEY
Labeling key:

0 or 1
Label contours with legend or color (default).

1
No contour labeling.

N
Same as 1 except show alphabetic legend only on every *N*th element.

Command Default

Show contour line labels.

Notes

Labels contours for identification with alphabetic legend for vector displays and color for raster displays. Number of contours is automatically reduced to 9 (or fewer) for clarity. Use **/CONTOUR** command to increase (24 maximum for alphabetic labeling; no limit for color labeling).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Contour Labeling

/CLEAR, *Read*

Clears the database.

DATABASE: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Read

File read option:

START

Reread **start81.ans** file (default).

NOSTART

Do not reread **start81.ans** file.

Notes

Resets the ANSYS database to the conditions at the beginning of the problem. Sets the import and Boolean options back to the ANSYS default. All items are deleted from the database and memory values are set to zero for items derived from database information. All files are left intact. This command is useful between multiple analyses in the same run, or between passes of a multipass analysis (such as between the substructure generation, use, and expansion passes). Should not be used in a do-loop since loop counters will be reset. The **start81.ans** file will be reread (by default) after the database is cleared, unless *Read* is set to NOSTART. Additional commands cannot be stacked (using the $ separator) on the same line as the **/CLEAR** command.

Use care when placing the **/CLEAR** command within branching constructs (for example, those employing ***DO** or ***IF** commands). The command deletes all parameters including the looping parameter for do-loops. (You can preserve your iteration parameter by issuing a **PARSAV** command prior to the **/CLEAR** command, then following the **/CLEAR** command with a **PARRES** command.)

/CLEAR resets the jobname to match the currently open session **.LOG** and **.ERR** files. This will return the jobname to its original value, or to the most recent value specified on **/FILNAME** with *KEY* = 1.

This command is valid only at the Begin level.

Menu Paths

Utility Menu>File>Clear & Start New

CLOCAL, *KCN*, *KCS*, *XL*, *YL*, *ZL*, *THXY*, *THYZ*, *THZX*, *PAR1*, *PAR2*

Defines a local coordinate system relative to the active coordinate system.

DATABASE: Coordinate System

MP ME ST DY <> PR EM <> FL PP ED

KCN

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

KCS

Coordinate system type:

0 or CART
Cartesian

1 or CYLIN
Cylindrical (circular or elliptical)

2 or SPHE
Spherical (or spheroidal)

3 or TORO
Toroidal

XL, *YL*, *ZL*
Location (in the active coordinate system) of the origin of the new coordinate system (R, θ, Z for cylindrical, R, θ,Φ for spherical or toroidal).

THXY
First rotation about local Z (positive X toward Y).

THYZ
Second rotation about local X (positive Y toward Z).

THZX
Third rotation about local Y (positive Z toward X).

PAR1
Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

PAR2
Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

Notes

Defines and activates a local coordinate system by origin location and orientation angles relative to the active coordinate system. This local system becomes the active coordinate system, and is automatically aligned with the active system (i.e., x is radial if a cylindrical system is active, etc.). Nonzero rotation angles (degrees) are relative to this automatic rotation. See the **CS**, **CSKP**, **CSWPLA**, and **LOCAL** commands for alternate definitions. Local coordinate systems may be displayed with the **/PSYMB** command.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

CLOG, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*, *FACTB*

Forms the common log of a variable

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA
Reference number of the variable to be operated on.

--, --
Unused fields.

Name
Thirty-two character name for identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --
Unused fields.

FACTA
Scaling factor applied to variable *IA* (defaults to 1.0).

FACTB
Scaling factor (positive or negative) applied to the operation (defaults to 1.0).

Notes

Forms the common log of a variable according to the operation:

IR = *FACTB**LOG(*FACTA* x *IA*)

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Common Log

/CLOG, *Fname*, *Ext*, --

Copies the session log file to a named file.

SESSION: Files

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path to which the log file is to be copied (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Ext
Filename extension (8 character maximum).

--
Unused field

Notes

This command is valid in any processor, but only during an interactive run.

Menu Paths

This command cannot be accessed from a menu.

CLRMSHLN
Clears meshed entities.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

Notes

When you use the GUI method to set the number of elements on specified lines, and any of those lines is connected to one or more meshed lines, areas, or volumes, ANSYS gives you the option to clear the meshed entities. (This occurs only when you perform this operation via the GUI; ANSYS does not provide such an option when you use the command method [**LESIZE**].)

If you activate the mesh clearing option, the program invokes an ANSYS macro, **CLRMSHLN**, that clears the meshed entities. This macro name will appear in the log file (**Jobname.LOG**). This macro is for the ANSYS program's internal use only. This command is not intended to be typed in directly in an ANSYS session, although it can be included in an input file for batch input or for use with the **/INPUT** command.

Menu Paths

This command cannot be accessed from a menu.

CM, *Cname*, *Entity*
Groups geometry items into a component.

DATABASE: Components

MP ME ST DY <> PR EM <> FL PP ED

Cname
An alphanumeric name used to identify this component. *Cname* may be up to 32 characters, beginning with a letter and containing only letters, numbers, and underscores. Component names beginning with an underscore (e.g., _LOOP) are reserved for use by ANSYS and should be avoided. Components named "ALL," "STAT," and "DEFA" are not permitted. Overwrites a previously defined name.

Entity
Label identifying the type of geometry items to be grouped:

VOLU
Volumes.

AREA
Areas.

LINE
Lines.

KP
Keypoints.

ELEM
Elements.

NODE
Nodes.

Notes

Components may be further grouped into assemblies [**CMGRP**]. The selected items of the specified entity type will be stored as the component. Use of this component in the select command [**CMSEL**] causes all these items to be selected at once, for convenience.

A component is a grouping of some geometric entity that can then be conveniently selected or unselected. A component may be redefined by reusing a previous component name. The following entity types may belong to a component: nodes, elements, keypoints, lines, areas, and volumes. A component may contain only 1 entity type, but an individual item of any entity may belong to any number of components. Once defined, the items contained in a component may then be easily selected or unselected [**CMSEL**]. Components may be listed [**CMLIST**], modified [**CMMOD**] and deleted [**CMDELE**]. Components may also be further grouped into assemblies [**CMGRP**]. Other entities associated with the entities in a component (e.g., the lines and keypoints associated with areas) may be selected by the **ALLSEL** command.

An item will be deleted from a component if it has been deleted by another operation (see the **KMODIF** command for an example). Components are automatically updated to reflect deletions of one or more of their items. Components are automatically deleted and a warning message is issued if all their items are deleted. Assemblies are also automatically updated to reflect deletions of one or more of their components or subassemblies, but are not deleted if all their components and subassemblies are deleted.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Element Components
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Section
Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts
Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Element Components
Main Menu>Solution>Define Loads>Delete>Structural>Section
Utility Menu>Select>Comp/Assembly>Create Component

/CMAP, *Fname*, *Ext*, *Dir*, *Kywrd*, *NCNTR*

Changes an existing or creates a new color mapping table.

GRAPHICS: Set Up
DISPLAY: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

If blank, restore color map.

Ext
Filename extension (8 character maximum).

Dir
Directory or path specification. You can use any portion of the 248 character limit (above).

Kywrd
Keyword indicating the disposition of the color map file.

(blank)
Loads existing color map file.

CREATE
Starts CMAP utility and modifies or creates the specified file.

SAVE
Writes the active color map to the specified file, which can be imported into future ANSYS sessions.

NCNTR
Number of contours to be defined by the **/CMAP** command (max = 128). If no value is specified, CMAP defaults to 9, even if an existing file is being modified.

Command Default

Use predefined ANSYS color map table.

Notes

Reads the color map file (RGB index specifications) to change from current specifications. Only one color map may be active at a time. See **/COLOR** for other color controls. Changing the color map in ANSYS with the **/CMAP** command will change the meaning of the color labels on the **/COLOR** command.

This command is valid anywhere.

Menu Paths

Utility Menu>PlotCtrls>Redirect Plots>To GRPH File
Utility Menu>PlotCtrls>Redirect Plots>To HPGL File
Utility Menu>PlotCtrls>Redirect Plots>To HPGL2 File
Utility Menu>PlotCtrls>Redirect Plots>To PSCR File
Utility Menu>PlotCtrls>Style>Colors>Default Color Map

CMATRIX, *SYMFAC*, *Condname*, *NUMCOND*, *GRNDKEY*, *Capname*

Performs electrostatic field solutions and calculates the self and mutual capacitances between multiple conductors.

SOLUTION: Analysis Options

MP ME ST <> <> <> EM <> <> PP ED

SYMFAC
Geometric symmetry factor. Capacitance values are scaled by this factor which represents the fraction of the total device modeled. Defaults to 1.

Condname
Alpha-numeric prefix identifier used in defining named conductor components.

NUMCOND
Total Number of Components. If a ground is modeled, it is to be included as a component. If a ground is not modeled, but infinite elements are used to model the far-field ground, a named component for the far-field ground is not required.

GRNDKEY
Ground key:

0
Ground is one of the components, which is not at infinity.

1
Ground is at infinity (modeled by infinite elements or a Trefftz domain).

Capname
Array name for computed capacitance matrix. Defaults to **CMATRIX**.

Notes

To invoke the **CMATRIX** macro, the exterior nodes of each conductor must be grouped into individual components using the **CM** command. Each set of independent components is assigned a component name with a common prefix followed by the conductor number. A conductor system with a ground must also include the ground nodes as a component. The ground component is numbered last in the component name sequence.

A Ground Capacitance matrix is a matrix relating charge to a voltage vector. A ground matrix can not be applied to a circuit modeler such as SPICE. The Lumped Capacitance matrix is a matrix formed by a combination of lumped "arrangements" of voltage differences between conductors. You can use the lumped capacitance terms in a circuit modeler to represent capacitances between conductors.

You must enclose all name-strings in single quotes in the **CMATRIX** command line.

See the *ANSYS, Inc. Theory Reference* and HMAGSOLV in the *ANSYS Low-Frequency Electromagnetic Analysis Guide* for details.

Menu Paths

Main Menu>Solution>Solve>Electromagnet>Static Analysis>Capac Matrix

CMDELE, *Name*

Deletes a component or assembly definition.

DATABASE: Components

MP ME ST DY <> PR EM <> FL PP ED

Name
Name of the component or assembly whose definition is to be removed.

Notes

Entities contained in the component, or the components within the assembly, are unaffected. Only the grouping relationships are deleted. Assemblies are automatically updated to reflect deletion of their components or subassemblies, but they are not automatically deleted when all their components or subassemblies are deleted.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Section
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Layers>Picked Lines
Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements
Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts
Main Menu>Solution>Define Loads>Delete>Structural>Section
Utility Menu>Select>Comp/Assembly>Delete Comp/Assembly

CMDOMEGA, *CM_NAME*, *DOMEGAX*, *DOMEGAY*, *DOMEGAZ*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

Specifies the rotational acceleration of an element component about a user-defined rotational axis.

SOLUTION: Inertia

MP ME ST <> <> PR <> <> <> PP ED

CM_NAME,
The name of the element component (8 character maximum).

DOMEGAX, *DOMEGAY*, *DOMEGAZ*
If the *X2*, *Y2*, *Z2* fields are not defined, *DOMEGAX*, *DOMEGAY*, and *DOMEGAZ* specify the components of the rotational acceleration vector in the global Cartesian X, Y, Z directions.

If the *X2*, *Y2*, *Z2* fields are defined, only *DOMEGAX* is required. *DOMEGAX* specifies the scalar rotational acceleration about the rotational axis. The rotational direction of *DOMEGAX*is designated either positive or negative, and is determined by the "right hand rule."

X1, *Y1*, *Z1*
If the *X2*, *Y2*, *Z2* fields are defined, *X1*, *Y1*, and *Z1* define the coordinates of the beginning point of the rotational axis vector. Otherwise, *X1*, *Y1*, and *Z1* are the coordinates of a point through which the rotational axis passes.

X2, *Y2*, *Z2*
The coordinates of the end point of the rotational axis vector.

Notes

Specifies the rotational acceleration components *DOMEGAX*, *DOMEGAY*, and *DOMEGAZ* of an element component *CM_NAME* about a user-defined rotational axis. The rotational axis can be defined either as a vector passing through a single point, or a vector connecting two points.

You can use the **CMDOMEGA** command to specify acceleration based loading on up to 100 rotational element components.

You can define rotational acceleration and rotational axis with the **CMDOMEGA** command for STATIC, HARMIC (full), TRANS (full), and SUBSTR analyses. Rotational velocities are combined with the element mass matrices to form a body force load vector term. Units are radians/time^2. Related commands are **ACEL**, **CGLOC**, **CGLOC**, **OMEGA**, **CMOMEGA**, **DCGOMG**, **DOMEGA**.

You can use the **CMDOMEGA** command in conjunction with any one of the following two groups of commands, but not with both groups simultaneously:

GROUP ONE: **OMEGA**, **DOMEGA**.
GROUP TWO: **CGOMGA**, **DCGOMG**, **CGLOC**.

Components for which rotational loading is to be specified must be made up of elements only. The elements you use cannot be part of more than one component, and elements that share nodes cannot exist in different element components.

See the *ANSYS, Inc. Theory Reference* for more information.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By Axis
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By origin
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpt
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpts
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Node
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Accel>On Component
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By Axis
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By origin
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpt

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpts
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Node
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Accel>On Component

---

CMEDIT, *Aname*, *Oper*, *Cnam1*, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*

Edits an existing assembly.

DATABASE: Components

MP ME ST DY <> PR EM <> FL PP ED

Aname
Name of the assembly to be edited.

Oper
Operation label:

ADD
To add more components. The level of any assembly to be added must be lower than that of the assembly *Aname* (see **CMGRP** command).

DELE
To remove components.

Cnam1, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*
Names of components and assemblies to be added to or deleted from the assembly.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Comp/Assembly>Edit Assembly

CMGRP, *Aname*, *Cnam1*, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*, *Cnam8*

Groups components and assemblies into an assembly.

DATABASE: Components

MP ME ST DY <> PR EM <> FL PP ED

Aname
An alphanumeric name used to identify this assembly. *Aname* may be up to 32 characters, beginning with a letter and containing only letters, numbers, and underscores. Overwrites a previously defined *Aname* (and removes it from higher level assemblies, if any).

Cnam1, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*, *Cnam8*
Names of existing components or other assemblies to be included in this assembly.

Notes

Groups components and other assemblies into an assembly identified by a name. **CMGRP** is used for the initial definition of an assembly. An assembly is used in the same manner as a component. Up to 5 levels of assemblies within assemblies may be used.

An assembly is a convenient grouping of previously defined components and other assemblies. Assemblies may contain components only, other assemblies, or any combination. A component may belong to any number of assemblies. Up to 5 levels of nested assemblies may be defined. Components and assemblies may be added to or deleted from an existing assembly by the **CMEDIT** command. Once defined, an assembly may be listed, deleted, selected, or unselected using the same commands as for a component. Assemblies are automatically updated to reflect deletions of one or more of their components or lower-level assemblies. Assemblies are not automatically deleted when all their components or subassemblies are deleted.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Comp/Assembly>Create Assembly

CMLIST, *Name*, *Key*, *Entity*

Lists the contents of a component or assembly.

DATABASE: Components

MP ME ST DY <> PR EM <> FL PP ED

Name

Name of the component or assembly to be listed (if blank, list all selected components and assemblies). If *Name* is specified, then *Entity* is ignored.

Key

Expansion key:

0 - Do not list individual entities in the component.
1 (or EXPA) - List individual entities in the component.

Entity

If *Name* is blank, then the following entity types can be specified:

VOLU - List the volume components only.
AREA - List the area components only.
LINE - List the line components only.
KP - List the keypoint components only.
ELEM - List the element components only.
NODE - List the node components only.

Notes

This command is valid in any processor. For components, it lists the type of geometric entity. For assemblies, it lists the components and/or assemblies that make up the assembly.

Examples of possible usage:

CMLIST - List all selected components.
CMLIST, , EXPA - List all selected components and for each component list the underlying entity ID's.
CMLIST, *Name* - List the specified component.
CMLIST, *Name*, EXPA - List specified component along with all underlying entity ID's.
CMLIST, , EXPA, *Entity* - List all selected components of specified entity type. For each component also list the underlying entity ID's.

Menu Paths

Utility Menu>List>Components
Utility Menu>List>Other>Components
Utility Menu>Select>Comp/Assembly>List Comp/Assembly

CMMOD, *Cname*, *Keyword*, *Value*

Modifies the specification of a component.

DATABASE: Components

MP ME ST DY <> PR EM <> FL PP ED

Cname
Name of the existing component or assembly to be modified.

Keyword
The label identifying the type of value to be modified.

NAME - Modify the NAME of the component

Value
If *Keyword* is NAME, then the value is the alphanumeric label to be applied. See the **CM** command for naming convention details. If a component named *Value* already exists, the command will be ignored and an error message will be generated.

Notes

The naming conventions for components, as specified in the **CM** command, apply for **CMMOD** (32 characters, "ALL", "STAT" and "DEFA" are not allowed, etc.). However, if you choose a component name that is already designated for another component, an error message will be issued and the command will be ignored.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

CMOMEGA, *CM_NAME*, *OMEGAX*, *OMEGAY*, *OMEGAZ*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*, *KSPIN*

Specifies the rotational velocity of an element component about a user-defined rotational axis.

SOLUTION: Inertia

MP ME ST <> <> PR <> <> <> PP ED

CM_NAME,
The name of the element component (8 character maximum).

OMEGAX, *OMEGAY*, *OMEGAZ*
If the *X2*, *Y2*, *Z2* fields are not defined, *OMEGAX*, *OMEGAY*, and *OMEGAZ* specify the components of the rotational velocity vector in the global Cartesian X, Y, Z directions.

If the *X2*, *Y2*, *Z2* fields are defined, only *OMEGAX* is required. *OMEGAX* specifies the scalar rotational velocity about the rotational axis. The rotational direction of *OMEGAX* is designated either positive or negative, and is determined by the "right hand rule."

X1, *Y1*, *Z1*
If the *X2*, *Y2*, *Z2* fields are defined,*X1*, *Y1*, and *Z1* define the coordinates of the beginning point of the rotational axis vector. Otherwise, *X1*, *Y1*, and *Z1* are the coordinates of a point through which the rotational axis passes.

X2, *Y2*, *Z2*
The coordinates of the end point of the rotational axis vector.

KSPIN
The spin softening key:

0 – No modification of the stiffness matrix of the element component due to rotation.
1 – Decrease radial stiffness of the element component due to rotation (i.e., include spin softening effects).

Notes

Specifies the rotational velocity components *OMEGAX*, *OMEGAY*, and *OMEGAZ* of an element component *CM_NAME* about a user-defined rotational axis. The rotational axis can be defined either as a vector passing through a single point or a vector connecting two points.

You can use the **CMOMEGA** command to specify velocity based loading on up to 100 rotational element components.

You define rotational velocity and rotational axis with the **CMOMEGA** command for STATIC, HARMIC (full), TRANS (full), and SUBSTR analyses. Rotational velocities are combined with the element mass matrices to form a body force load vector term. Units are radians/time. Related commands are **ACEL**, **CGLOC**, **CGLOC**, **CGOMGA**, **CMDOMEGA**, **DCGOMG**, **DOMEGA**.

You can use the **CMOMEGA** command in conjunction with any one of the following two groups of commands, but not with both groups simultaneously:

GROUP ONE: **OMEGA**, **DOMEGA**.
GROUP TWO: **CGOMGA**, **DCGOMG**, **CGLOC**.

Components for which rotational loading is to be specified must be made up of elements only. The elements you use cannot be part of more than one component, and elements that share nodes cannot exist in different element components.

The *KSPIN* option allows adjusting the stiffness of a rotating body to account for dynamic mass effects. The adjustment is called *spin-softening* and applies to a modal (**ANTYPE**,MODAL) or harmonic (**ANTYPE**,HARM) analysis only. The adjustment approximates the effects of geometry changes caused by large-deflection circumferential motion in a small-deflection analysis. The *KSPIN* option is not intended for a large-deflection static analysis; in such a case, use the **NLGEOM**,ON command to account for this effect. The command is usually used in conjunction with prestressing (**PSTRES**).

If your **CMOMEGA** parameters change, you should not reuse the overall stiffness matrix. See the *ANSYS, Inc. Theory Reference* for more information on the stiffness matrix.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By Axis
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By origin
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpt
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpts
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Node
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Veloc>On Component
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By Axis
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By origin
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpt
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpts
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Node
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Veloc>On Component

---

CMPLOT, *Label*, *Entity*, *Keyword*

Plots the entities contained in a component or assembly.

DATABASE: Components

MP ME ST DY <> PR EM <> FL PP ED

Label

Name of the component or assembly to be plotted.

(blank)
All selected components and assemblies are plotted (default). If fewer than 11 components are selected, then all are plotted. If more than 11 components are selected, then only the first 11 are plotted.

ALL
All selected components are plotted. If number of selected components is greater than 11, then the legend showing component names will not be shown.

N
Next set of defined components and assemblies is plotted.

P
Previous set of defined components and assemblies is plotted.

Cname
The specified component or assembly is plotted.

Set No.
The specified set number is plotted.

Entity
If *Label* is BLANK or ALL, then the following entity types can be specified:

VOLU - Plot the volume components only.
AREA - Plot the area components only.
LINE - Plot the line components only.
KP - Plot the keypoint components only.
ELEM - Plot the element components only.
NODE - Plot the node components only.

Keyword
For *Keyword* = ALL, plot the specified component name in the *Label* field in the context of all entities of the same type. Not valid if *Label* field is BLANK or ALL.

Notes

Components are plotted with their native entities. For assemblies, all native entities for the underlying component types are plotted simultaneously. Although more components can be plotted, the legend displays only 11 at a time. When more than eleven are plotted, the legend is not displayed.

Possible usage:

CMPLOT, *Cname* - Plots the specified component (if selected).
CMPLOT, *Cname*, ALL - Plot component in the context of all other selected entity components of the same type as the component.
CMPLOT - Plot the first eleven selected components.
CMPLOT, ALL - Plot all selected components.
CMPLOT, N or *CMPLOT*, P - next or previous set of eleven components.
CMPLOT, ALL, Entity - Plot all selected components of type specified in Entity.
CMPLOT, , Entity - Plot components of type specified in Entity, from the first eleven components.
CMPLOT, N, Entity - Plot components of type specified in Entity, if any, from the next set of eleven components (substitute P for N to plot from previous set).

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Components>By Name / Set Number
Utility Menu>Plot>Components>Next Set
Utility Menu>Plot>Components>Previous Set
Utility Menu>Plot>Components>Selected Components

CMSEL, *Type*, *Name*, *Entity*

Selects a subset of components and assemblies.

DATABASE: Components

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

ALL
Also select all components.

NONE
Unselect all components.

Name
Name of component or assembly whose items are to be selected (valid only if *Type* = S, R, A, or U).

Graphical picking is enabled if *Type* is blank and *Name* = P.

Entity
If *Name* is blank, then the following entity types can be specified:

VOLU - Select the volume components only.
AREA - Select the area components only.
LINE - Select the line components only.
KP - Select the keypoint components only.
ELEM - Select the element components only.
NODE - Select the node components only.

Notes

Selecting by component is a convenient adjunct to individual item selection (e.g., **VSEL**, **ESEL**, etc.). **CMSEL**, ALL allows you to select components **in addition** to other items you have already selected.

If *Type* = R for an assembly selection [**CMSEL**,R,<*assembly-name*>], the reselect operation is performed on each component in the assembly in the order in which the components make up the assembly. Thus, if one reselect operation results in an empty set, subsequent operations will also result in empty sets. For example, if the first reselect operation tries to reselect node 1 from the selected set of nodes 3, 4, and 5, the operation results in an empty set (that is, no nodes are selected). Since the current set is now an empty set, if the second reselect operation tries to reselect any nodes, the second operation also results in an empty set, and so on. This is equivalent to repeating the command **CMSEL**,R,<*component-name*> once for each component making up the assembly.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Joint Elems
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Joint Elems
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Joint Elems
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Joint Elems
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Section
Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements
Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Joint Elems
Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Joint Elems
Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Joint Elems
Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Joint Elems
Main Menu>Solution>Define Loads>Delete>Structural>Section
Utility Menu>Select>Comp/Assembly>Select Comp/Assembly

CMSFILE, *OPTION*, *Fname*, *Ext*

Specifies a list of component mode synthesis (CMS) results files for plotting results on the assembly.

POST1: Special Purpose

<> ME ST <> <> <> <> <> PP ED

OPTION

Specifies the command operation:

ADD
: Add the specified component results file (*Fname*) to the list of files to plot. This option is the default.

DELETE
: Remove the specified component results file (*Fname*) from the list of files to plot.

LIST
: List all specified component results file(s).

CLEAR
: Clear all previous files added.

ALL
: Add *all* component results (**.rst**) files from the working directory to the list of files to plot.

Fname
The file name (with full directory path) of the component results file. The default file name is the **Jobname** (specified via the **/FILNAME** command).

Ext
The file name (*Fname*) extension. The default extension is **.rst**.

Command Default

If issued with no arguments, the **CMSFILE** command uses these defaults:

CMSFILE,ADD,*Jobname*,rst

The command adds the component results file **Jobname.rst**.

Notes

The **CMSFILE** command specifies the list of component mode synthesis (CMS) results files to include when plotting the mode shape of an assembly.

During postprocessing (**/POST1**) of a CMS analysis, issue the **CMSFILE** command to point to component results files of interest. (You can issue the command as often as needed to include all or some of the component results files.) Issue the **SET** command to acquire the frequencies and mode shapes from substeps for all specified results files. Execute a plot (**PLNSOL**) or print (**PRNSOL**) operation to display the mode shape of the entire assembly.

When you specify a results file to add to the plot list, the ANSYS program first verifies that the file is from a CMS analysis and that the frequencies of the result sets on the file match the frequencies on the first file in the list.

If *OPTION* = DELETE or CLEAR, you must clear the database, then re-enter the postprocessor (**/POST1**) and issue a **SET** command for the change to take effect on subsequent plots.

Clearing the database does *not* clear the list of files specified via the **CMSFILE** command. Specify *OPTION* = CLEAR to clear the list of files.

Menu Paths

Main Menu>General Postproc>Data & File Opts

CMSOPT, *CMSMETH*, *NMODE*, *FREQB*, *FREQE*, *FBDDEF*, *FBDVAL*

Specifies component mode synthesis (CMS) analysis options.

SOLUTION: Analysis Options

<> ME ST <> <> <> <> <> PP ED

CMSMETH
The component mode synthesis method to use. This value is required.

FIX
Fixed-interface method.

FREE
Free-interface method.

NMODE
The number of normal modes extracted and used in superelement generation. This value is required and the minimum is 1.

FREQB
Beginning, or lower end, of frequency range of interest. This value is optional.

FREQE
Ending, or upper end, of frequency range of interest. This value is optional.

FBDDEF
In a free-interface CMS analysis (*CMSMETH* = FREE), the method to use for defining free body modes:

FNUM
The number (*FDBVAL*) of rigid body modes in the calculation.

FTOL
Employ a specified tolerance (*FDBVAL*) to determine rigid body modes in the calculation.

FAUTO
Automatically determine rigid body modes in the calculation. This method is the default.

RIGID
If no rigid body modes exist, define your own via the **RIGID** command.

FBDVAL
In a free-interface CMS analysis (*CMSMETH* = FREE), the number of rigid body modes if *FBDDEF* = FNUM (where the value is an integer from 0 through 6), *or* the tolerance to employ if *FBDDEF* = FTOL (where the value is a positive real number representing rad/sec). This value is required only when *FBDDEF* = FNUM or *FBDDEF* = FTOL; otherwise, any specified value is ignored.

Command Default

Issuing the **CMSOPT** command with no arguments is invalid. You must specify at least the CMS method (*CMSMETH*) and the number of modes (*NMODE*). In a free-interface CMS analysis (*CMSMETH* = FREE), the default method for determining rigid body modes is FAUTO (automatic).

Notes

CMS employs the Block Lanczos eigensolution method in the generation pass.

CMS does not yet support damping matrix reduction. ANSYS sets the matrix generation key to 2 automatically (**SEOPT**,*SEMATR*).

CMS does not support the **SEOPT**,,,,,RESOLVE command. Instead, ANSYS sets the expansion method for the expansion pass (*EXPMTH*) to BACKSUB.

This command is also valid in **/PREP7**.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

CNCHECK, *Option*, *RID1*, *RID2*, *RINC*

Provides the initial status of contact pairs.

PREP7: Database
SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> FL PP ED

Option
Option to be performed:

DETAIL
List all contact pair properties (default).

SUMMARY
List only the open/closed status for each contact pair.

POST
Execute a partial solution to write the initial contact configuration to a results file, **Jobname.RCN**.

ADJUST
Physically move contact nodes to target to close gap or reduce penetration.

RID1, *RID2*, *RINC*
The range of real constant pair ID's for which *Option* will be performed. If *RID2* is not specified, it defaults to *RID1*. If no value is specified, all contact pairs in the selected set of elements are considered.

Notes

CNCHECK provides information for surface-to-surface and node-to-surface contact pairs (element types TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, CONTA174, CONTA175). All contact and target elements of interest, along with the solid elements and nodes attached to them, must be selected for the command to function properly.

CNCHECK is available both in PREP7 and in SOLUTION, but only before the first SOLVE, that is, only before the first load step or the first substep.

The command **CNCHECK**,POST solves the initial contact configuration in one substep. After issuing this command, you can postprocess the contact result items as you would for any other converged load step. However, only the contact status, contact penetration or gap, and contact pressure will have meaningful values. Other contact quantities (friction stress, sliding distance, chattering) will be available, but not useful.

If **CNCHECK**,POST is issued within the SOLUTION processor, the **SOLVE** command that solves the first load step of your analysis should appear in a different step, as shown in the following example:

```
/SOLU
CNCHECK,POST
FINISH
. . .

/SOLU
SOLVE
FINISH
. . .
```

CNCHECK,POST writes initial contact results to a results file named **Jobname.RCN**. When postprocessing the initial contact state, you need to explicitly read results from this file using the **FILE** and **SET**,FIRST commands in

POST1 to properly read the corresponding contact data. Otherwise, the results file may be read improperly. The following example shows a valid command sequence for plotting the initial contact gap:

```
/SOLU
CNCHECK,POST
FINISH
/POST1
FILE,Jobname,RCN
SET,FIRST
PLNSOL,CONT,GAP,0,1
FINISH
. . .
```

You can issue **CNCHECK**,ADJUST to physically move contact nodes to the target surface. See Section 11.4.8.8: Physically Moving Contact Nodes Towards the Target Surface in the *ANSYS Structural Analysis Guide* for more information. Similar to the POST option described above, if **CNCHECK**,ADJUST is issued within the SOLUTION processor, the **SOLVE** command that solves the first load step of your analysis should appear in a different step:

```
/SOLU
CNCHECK,ADJUST
FINISH
. . .

/SOLU
SOLVE
FINISH
. . .
```

For performance reasons, ANSYS internally uses a subset of nodes and elements based on the specified contact regions (*RID1*, *RID2*, *RINC*) when executing **CNCHECK**,POST or **CNCHECK**,ADJUST.

Menu Paths

Main Menu> Preprocessor> Modeling> Create> Contact Pair

CNVTOL, *Lab*, *VALUE*, *TOLER*, *NORM*, *MINREF*

Sets convergence values for nonlinear analyses.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR EM <> <> PP ED

Lab

Valid convergence labels are as follows: If STAT, list the status of the currently specified criteria. Structural labels: U (displacements); ROT (rotations); F (forces); M (moments). Thermal labels: TEMP (temperature); HEAT (heat flow). Fluid labels: PRES (pressures); V (velocities); FLOW (fluid flow); VF (fluid force). Electric labels: VOLT (voltage); EMF (electromotive force); CURR (current flow); AMPS (current flow); CURT (current flow). Magnetic labels: MAG (scalar magnetic potential); A (vector magnetic potentials); CURR (current flow); FLUX (scalar magnetic flux); CSG (magnetic current segments); VLTG (voltage drop).

VALUE

Typical value for the above label for this analysis. If negative, and if this convergence label was previously specified explicitly, then convergence based on this label is removed. (A negative *VALUE* will not remove a default convergence label.) Defaults to the maximum of a program calculated reference or *MINREF*. For degrees of freedom, the reference is based upon the selected *NORM* and the current total DOF value. For forcing quantities, the reference is based upon the selected *NORM* and the applied loads.

TOLER
When **SOLCONTROL**,ON, tolerance about *VALUE*. Defaults to 0.005 (0.5%) for force and moment, and 0.05 (5%) for displacement when rotational DOFs are not present. When **SOLCONTROL**,OFF, defaults to 0.001 (0.1%) for force and moment.

NORM
Specifies norm selection:

2
L2 norm (check SRSS value) (default).

1
L1 norm (check absolute value sum).

0
Infinite norm (check each DOF separately).

MINREF
The minimum value allowed for the program calculated reference value. If negative, no minimum is enforced. Used only if *VALUE* is blank. Defaults to 0.01 for force and moment convergence, 1.0E-6 for heat flow, 1.0E-12 for VOLT and AMPS, and 0.0 otherwise. When **SOLCONTROL**,OFF, defaults to 1.0 for force and moment convergence. The default for heat flow (1.0E-6), VOLT and AMPS (1.0E-12), and others are independent of the **SOLCONTROL** setting.

Command Default

For static or transient analysis, check the out-of-balance load for any active DOF using the default *VALUE*, *TOLER*, *NORM*, and *MINREF*. Also check the displacement convergence for some problems. For harmonic magnetic analysis, check the out-of-balance of the degrees of freedom.

Notes

The default values given for this command assume **SOLCONTROL**,ON (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL**,ON and **SOLCONTROL**,OFF.

Values may be set for the degrees of freedom (DOF) and/or the out-of-balance load for the corresponding forcing quantities. When the GUI is on, if a "Delete" operation in a **Nonlinear Convergence Criteria** dialog box writes this command to a log file (**Jobname.LOG** or **Jobname.LGW**), you will observe that *Lab* is blank, *VALUE* = -1, and *TOLER* is an integer number. In this case, the GUI has assigned a value of *TOLER* that corresponds to the location of a chosen convergence label in the dialog box's list. It is *not* intended that you type in such a location value for *TOLER* in an ANSYS session. However, a file that contains a GUI-generated **CNVTOL** command of this form can be used for batch input or with the **/INPUT** command.

This command is also valid in PREP7.

Convergence norms specified with **CNVTOL** may be graphically tracked while the solution is in process using the ANSYS program's Graphical Solution Tracking (GST) feature. Use the **/GST** command to turn GST on or off. By default, GST is ON for interactive sessions and OFF for batch runs.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Convergence Crit

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Harmonic
Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Static
Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Transient
Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Solution>Load Step Opts>Nonlinear>Convergence Crit
Main Menu>Solution>Load Step Opts>Nonlinear>Harmonic
Main Menu>Solution>Load Step Opts>Nonlinear>Static
Main Menu>Solution>Load Step Opts>Nonlinear>Transient

/COLOR, *Lab*, *Clab*, *N1*, *N2*, *NINC*

Specifies the color mapping for various items.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Lab

Apply color to the items specified by the following labels:

AXES
Determines the color (specified in next argument, *Clab*) that the axes of a graph will be plotted in.

AXNUM
Determines the color (specified in next argument, *Clab*) that the numbering on the axes of a graph will be plotted in.

NUM
Discretely numbered items (such as element types, element materials, etc., as shown on the **/PNUM** command). Also specify number (1 to 11) in the *N1* field. For example, **/COLOR**,NUM,RED,3 will assign the color red to all items having the discrete number 3 (material displays would show elements having material 3 as red).

OUTL
Outline of elements, areas, and volumes. Ex: **/COLOR**,OUTL,BLUE.

ELEM
Elements. Use *N1*, *N2*, *NINC* fields for element numbers.

LINE
Solid model lines. Use *N1*, *N2*, *NINC* fields for line numbers.

AREA
Solid model areas. Use *N1*, *N2*, *NINC* fields for area numbers.

VOLU
Solid model volumes. Use *N1*, *N2*, *NINC* fields for volume numbers.

ISURF
Isosurfaces (surfaces of constant stress, etc.). This option is particularly useful when capturing frames for animating a single isosurface value.

WBAK
Window background. Use *N1*, *N2*, *NINC* fields for window numbers. The options that you select using *Lab* = PBAK will supersede those applied using *Lab* = WBAK.

b.c.label --
Boundary condition label. Enter U, ROT, TEMP, PRES, V, ENKE, ENDS, SP01 through SP06 or their user-defined names, VOLT, MAG, A, EMF, CURR, F, M, HEAT, FLOW, VF, AMPS, FLUX, CSG, CURT, VLTG, MAST, CP, CE, NFOR, NMOM, RFOR, RMOM, PATH. See the **/PBC** command for boundary condition label definitions.

GRBAK
Graph background.

GRID
Graph grid lines.

AXLAB
Graph X and Y axis labels.

CURVE
Graph curves (identify curve numbers (1-6) in *N1*, *N2*, *NINC* fields).

CM
Component group. Use *N1* field for component name, ignore *N2* and *NINC*.

CNTR
ANSYS contour stress colors. The maximum number of contours available is 128. The number of colors that can be specified interactively (GUI) is 9. (**/CONTOUR**, , 9). Any other setting will yield inconsistent results.

SMAX
Specifies that all stress values above the maximum value entered in **/CONTOUR** will be displayed in the color designated in the *Clab* field. Defaults to dark grey.

SMIN
Specifies that all stress values below the minimum value entered in **/CONTOUR** will be displayed in the color designated in the *Clab* field. Defaults to dark grey.

PBAK
Activates background shading options (see command syntax at end of argument descriptions below). The options that you select using *Lab* = PBAK will supersede those applied using *Lab* = WBAK.

Clab
Valid color labels are:

BLAC (0)
Black

MRED (1)
Magenta-Red

MAGE (2)
Magenta

BMAG (3)
Blue-Magenta

BLUE (4)
Blue

CBLU (5)
Cyan-Blue

CYAN (6)
Cyan

GCYA ((7)
Green-Cyan

GREE (8)
Green

YGRE (9)
Yellow-Green

YELL (10)
Yellow

ORAN (11)
Orange

RED (12)
Red

DGRA (13)
Dark Gray

LGRA (14)
Light Gray

WHIT (15)
White

N1, *N2*, *NINC*
Apply color to *Lab* items numbered *N1* to *N2* (defaults to *N1*) in steps of *NINC* (defaults to 1). If *N1* is blank, apply color to entire selected range. If *Lab* is CM, use component name for *N1* and ignore *N2* and *NINC*. If *N1* = P, graphical picking of elements, lines, areas and volumes is enabled; your can assign colors to the entities via the picker. When picking is enabled, the *Lab* and *Clab* fields are ignored.

If *Lab* = PBAK, the command format is **/COLOR**, PBAK,*Key_On_Off*, *KEY_TYPE*, *KEY_INDEX*.

The options that you select using *Lab* = PBAK will supersede those applied using *Lab* = WBAK.

Key_On_Off
Turns the background colors on and off. Acceptable values are ON (1) and OFF (0).

KEY_TYPE
Determines the type of background. Acceptable values are 0 (smooth shading left to right), 1 (smooth shading top to bottom), 2 (smooth shading right to left), 3 (smooth shading bottom to top), and -1 (textured image background)

KEY_INDEX
An integer value that corresponds to a background color or texture. If *Key_Type* is -1, the background will correspond to values specified in the **/TXTRE** command. If *Key_Type* is any other acceptable value, the background will correspond to the color values listed above under *Clab*.

Command Default

Use the default color mapping.

Notes

Issue **/COLOR**,STAT to display the current color mapping. Issue **/COLOR**,DEFA to reset the default color mapping.

Note — Color labels may also be reassigned any "color" with the **/CMAP** command.

The default background display for the ANSYS Graphics Window is **/COLOR**,PBAK,ON,1,BLACK. See the **/UIS** command for information on modifying your default values.

This command is valid anywhere.

Menu Paths

Utility Menu>PlotCtrls>Style>Colors>BC Colors
Utility Menu>PlotCtrls>Style>Colors>Component Colors
Utility Menu>PlotCtrls>Style>Colors>Entity Colors
Utility Menu>PlotCtrls>Style>Colors>Graph Colors
Utility Menu>PlotCtrls>Style>Colors>Numbered Item Colors
Utility Menu>PlotCtrls>Style>Colors>Window Colors

/COM, *Comment*

Places a comment in the output.

SESSION: List Controls

MP ME ST DY <> PR EM <> FL PP ED

Comment
Comment string, up to 75 characters.

Notes

The output from this command consists of the comment string. This command is similar to **C*** except that the comment produced by **C*** is more easily identified in the output. Parameter substitution within the comment occurs for every valid expression delimited by percent (%) signs. Enclosing such an expression in single quotes prevents parameter substitution.

Another way to include a comment is to precede it with a ! character (on the same line). The ! may be placed anywhere on the line, and any input following it is ignored as a comment. No output is produced by such a comment, but the comment line is included on the log file. This is a convenient way to annotate the log file.

This command is valid anywhere.

Menu Paths

This command cannot be accessed from a menu.

COMPRESS

Deletes all specified sets.

AUX3: Results Files

MP ME ST DY <> PR EM <> FL PP ED

Notes

Issue this command to delete all sets specified with the **DELETE** command.

Menu Paths

This command cannot be accessed from a menu.

CON4, *XCENTER*, *YCENTER*, *RAD1*, *RAD2*, *DEPTH*

Creates a conical volume anywhere on the working plane.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

XCENTER, *YCENTER*
Working plane X and Y coordinates of the center axis of the cone.

RAD1, *RAD2*
Radii of the faces of the cone. *RAD1* defines the bottom face and will be located on the working plane. *RAD2* defines the top face and is parallel to the working plane. A value of zero or blank for either *RAD1* or *RAD2* defines a degenerate face at the center axis (i.e., the vertex of the cone). The same value for both *RAD1* and *RAD2* defines a cylinder instead of a cone.

DEPTH
The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the cone. *DEPTH* cannot be zero (see Notes below).

Notes

Defines a solid conical volume with either the vertex or a face anywhere on the working plane. The cone must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) The face or faces will be circular (each area defined with four lines), and they will be connected with two areas (each spanning 180°). See the **CONE** command for an alternate way to create cones.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Primitives>Cone
Main Menu>Preprocessor>Modeling>Create>Volumes>Cone>By Picking

---

CONE, *RBOT*, *RTOP*, *Z1*, *Z2*, *THETA1*, *THETA2*

Creates a conical volume centered about the working plane origin.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

RBOT, *RTOP*

Radii of the bottom and top faces of the cone. A value of zero or blank for either *RBOT* or *RTOP* defines a degenerate face at the center axis (i.e., the vertex of the cone). The same value for both *RBOT* and *RTOP* defines a cylinder instead of a cone.

Z1, *Z2*

Working plane Z coordinates of the cone. The smaller value is always associated with the bottom face.

THETA1, *THETA2*

Starting and ending angles (either order) of the cone. Used for creating a conical sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0° and the ending angle defaults to 360°. See the *ANSYS Modeling and Meshing Guide* for an illustration.

Notes

Defines a solid conical volume centered about the working plane origin. The non-degenerate face (top or bottom) is parallel to the working plane but not necessarily coplanar with (i.e., "on") the working plane. The cone must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) For a cone of 360°, top and bottom faces will be circular (each area defined with four lines), and they will be connected with two areas (each spanning 180°). See the **CON4** command for an alternate way to create cones.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Cone>By Dimensions

---

/CONFIG, *Lab*, *VALUE*

Assigns values to ANSYS configuration parameters.

SESSION: Run Controls

MP ME ST DY <> PR EM EH FL PP <>

Lab

Configuration parameter to be changed:

NRES

VALUE is maximum number of results sets (substeps) allowed on the result file. Defaults to 1000. Minimum is 10.

NORSTGM

Option to write or not write geometry data to the results file. *VALUE* is either 0 (write geometry data) or 1 (do not write geometry data). Useful when complex analyses will create abnormally large files. Default is 0.

NBUF

VALUE is the number of buffers (1 to 32) per file in the solver. Defaults to 4.

NPROC
VALUE is the number of processors (system dependent) to use. Defaults to 1. If *VALUE* exceeds the number of available processors, ANSYS uses *all* available processors. ANSYS recommends setting *VALUE* no higher than the number of available processors *minus one*; for example, on a four-processor system, set *VALUE* to 3.

LOCFL
File open and close actions. For *VALUE* use: 0 for global (default); 1 for local. Applicable to **File.EROT**, **File.ESAV**, **File.EMAT**, and **File.TRI**. Typically used for large problems where locally closed files may be deleted earlier in the run with the **/FDELE** command.

SZBIO
VALUE is the record size (1024 to 4194304) of binary files (in integer words). Defaults to 16384 (system dependent).

ORDER
Automatic reordering scheme. For *VALUE* use: 0 for **WSORT**,ALL; 1 for **WAVES**; 2 for both **WSORT**,ALL and **WAVES** (default).

FSPLIT
Defines split points for binary files. *VALUE* is the file split point in megawords and defaults to the maximum file size for the system.

MXND
Maximum number of nodes. If not specified, defaults to 100 at first encounter. Dynamically expanded by doubling, even at first encounter, when maximum is exceeded.

MXEL
Maximum number of elements. Default and expansion as for MXND.

MXKP
Maximum number of keypoints. Default and expansion as for MXND.

MXLS
Maximum number of lines. Default and expansion as for MXND.

MXAR
Maximum number of areas. Default and expansion as for MXND.

MXVL
Maximum number of volumes. Default and expansion as for MXND.

MXRL
Maximum number of sets of real constants (element attributes). Default and expansion as for MXND.

MXCP
Maximum number of sets of coupled degrees of freedom. Default and expansion as for MXND.

MXCE
Maximum number of constraint equations. Default and expansion as for MXND.

NOELDB
Option to write or not write results into the database after a solution. When *VALUE* = 0 (default), write results into the database. When *VALUE* = 1, do not write results into the database.

DYNA_DBL
Option to invoke the double precision version of the explicit dynamics solver LS-DYNA. When *VALUE* = 0 (default), the single precision version is used. When *VALUE* = 1, the double precision version is used.

STAT
Displays current values set by the **/CONFIG** command.

VALUE
Value (an integer number) assigned to the configuration parameter.

Notes

All configuration parameters have initial defaults, which in most cases do not need to be changed. Where a specially configured version of the ANSYS program is desired, the parameters may be changed with this command. Issue **/CONFIG**,STAT to display current values. Changes must be defined before the parameter is required. These changes (and others) may also be incorporated into the config81.ans file which is read upon execution of the program (see The Configuration File in the *ANSYS Basic Analysis Guide*). If the same configuration parameter appears in both the configuration file and this command, this command overrides.

The **/CONFIG** command is not valid for the Multiphysics 1, 2, or 3 products.

Menu Paths

Utility Menu>List>Status>Configuration

CONJUG, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*

Forms the complex conjugate of a variable.

POST26: Operations

MP ME ST DY <> PR EM <> <> PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA
Reference number of the variable to be operated on.

--, --
Unused fields.

Name
Thirty-two character name for identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --
Unused fields.

FACTA
Scaling factor (positive or negative) applied to variable (default to 1.0).

Notes

Used only with harmonic analyses (**ANTYPE**,HARMIC).

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Complex Conjugat

/CONTOUR, *WN*, *NCONT*, *VMIN*, *VINC*, *VMAX*

Specifies the uniform contour values on stress displays.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

NCONT
Number of contour values. *NCONT* defaults to 9 for X11 or WIN32 and to 128 for X11c or WIN32C. The default graphics window display for 3-D devices is a smooth continuous shading effect that spans the maximum of 128 contours available. The legend, however, will display only nine color boxes, which span the full range of colors displayed in the graphics window.

VMIN
Minimum contour value. If *VMIN* = AUTO, automatically calculate contour values based upon *NCONT* uniformly spaced values over the min-max extreme range. Or, if *VMIN* = USER, set contour values to those of the last display (useful when last display automatically calculated contours).

VINC
Value increment (positive) between contour values. Defaults to (*VMAX*-*VMIN*)/*NCONT*.

VMAX
Maximum contour value. Ignored if both *VMIN* and *VINC* are specified.

Command Default

Nine contour values uniformly spaced between the extreme values, or no contours if the ratio of range to minimum value (or range to maximum if minimum = 0) is less than 0.001.

Notes

See the **/CVAL** command for alternate specifications. Values represent contour lines in vector mode, and the algebraic maximum of contour bands in raster mode.

Note — No matter how many contours (*NCONT*) are specified by **/CONTOUR**, the actual number of contours that appear on your display depends also on the device name, whether the display is directed to the screen or to a file, the display mode (vector or raster), and the number of color planes. (All these items are controlled by **/SHOW** settings.) In any case, regardless of whether they are smoothed or banded, only 128 contours can be displayed. See Creating Geometric Results Displays in the *ANSYS Basic Analysis Guide* for more information on changing the number of contours.

If the current ANSYS graphics are *not* displayed as Multi-Plots, then the following is true: If the current device is a 3-D device [**/SHOW**,3D], the model contours in all active windows will be the same, even if separate **/CONTOUR** commands are issued for each active window. For efficiency, ANSYS 3-D graphics logic maintains a single data structure (segment), which contains precisely one set of contours. The program displays the same segment in all windows. The view settings of each window constitute the only differences in the contour plots in the active windows.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Uniform Contours

/COPY, *Fname1*, *Ext1*, --, *Fname2*, *Ext2*, --
Copies a file.

SESSION: Files

MP ME ST DY <> PR EM <> FL PP ED

Fname1
File name to be copied and its directory path (248 characters maximum for both file name and directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to the current **Jobname**.

Ext1
Filename extension (8 character maximum).

--
Unused field

Fname2
File name to be created and its directory path (248 characters maximum for both file name and directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Fname2 defaults to *Fname1*.

Ext2
Filename extension (8 character maximum).

Ext2 defaults to *Ext1*.

--
Unused field

Notes

The original file is untouched. Ex: **/COPY**,A,,,B copies file A to B in the same directory. **/COPY**,A,DAT,,,INP copies the file **A.DAT** to **A.INP**. See the *ANSYS Operations Guide* for details. ANSYS binary and ASCII files can be copied.

Menu Paths

Utility Menu>File>File Operations>Copy

COUPLE

Specifies "Node coupling" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Coupled Sets

COVAL, *TBLNO1*, *TBLNO2*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*

Defines PSD cospectral values.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

TBLN01
First input PSD table number associated with this spectrum.

TBLN02
Second input PSD table number associated with this spectrum.

SV1, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*
PSD cospectral values corresponding to the frequency points [**PSDFRQ**].

Notes

Defines PSD cospectral values to be associated with the previously defined frequency points. Two table references are required since values are off-diagonal terms. Unlike autospectra [**PSDVAL**], the cospectra can be positive or negative. The cospectral curve segment where there is a sign change is interpolated linearly (the rest of the curve segments use log-log interpolation). For better accuracy, choose as small a curve segment as possible wherever a sign change occurs.

Repeat **COVAL** command using the same table numbers for additional points (50 maximum per curve). This command is valid for **SPOPT**,PSD only.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Cospectral
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Cospectral

---

CP, *NSET*, *Lab*, *NODE1*, *NODE2*, *NODE3*, *NODE4*, *NODE5*, *NODE6*, *NODE7*, *NODE8*, *NODE9*, *NODE10*, *NODE11*, *NODE12*, *NODE13*, *NODE14*, *NODE15*, *NODE16*, *NODE17*

Defines (or modifies) a set of coupled degrees of freedom.

PREP7: Coupled DOF

MP ME ST DY <> PR EM <> FL PP ED

NSET

Set reference number:

n

Arbitrary set number.

HIGH

The highest defined coupled set number will be used (default, unless *Lab* = ALL). This option is useful when adding nodes to an existing set.

NEXT

The highest defined coupled set number plus one will be used (default if *Lab* = ALL). This option automatically numbers coupled sets so that existing sets are not modified.

Lab

Degree of freedom label for coupled nodes (in the nodal coordinate system). Defaults to label previously defined with *NSET* if set *NSET* already exists. A different label redefines the previous label associated with *NSET*. Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations) (in radians). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Fluid labels: PRES (pressure); VX, VY, or VZ (velocities). Electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials); CURR (current). Explicit analysis labels: UX, UY, or UZ (displacements). If *Lab* = ALL, sets will be generated for each active degree of freedom (i.e., one set for the UX degree of freedom, another set for UY etc.), and *NSET* will be automatically incremented to prevent overwriting existing sets. The ALL option cannot be used to modify existing sets--*NSET* must be a new set number *n* or NEXT. The degree of freedom set is determined from all element types defined and the **DOF** command, if used. ALL is the only label applicable to FLOTRAN.

NODE1, *NODE2*, *NODE3*, *NODE4*, *NODE5*, *NODE6*, *NODE7*, *NODE8*, *NODE9*, *NODE10*, *NODE11*, *NODE12*, *NODE13*, *NODE14*, *NODE15*, *NODE16*, *NODE17*

List of nodes to be included in set. Duplicate nodes are ignored. If a node number is input as negative, the node is deleted from the coupled set. The first node in the list is the primary (retained) node. If *NODE1* = ALL, *NODE2* through *NODE17* are ignored and all selected nodes [**NSEL**] are included in the set. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1*.

Notes

Do not include the same degree of freedom in more than one coupled set. Repeat **CP** command for additional nodes.

Coupling degrees of freedom into a set causes the results calculated for one member of the set to be the same for all members of the set. Coupling can be used to model various joint and hinge effects. A more general form of coupling can be done with constraint equations [**CE**]. For structural analyses, a list of nodes is defined along with the nodal directions in which these nodes are to be coupled. As a result of this coupling, these nodes are forced to take the same displacement in the specified *nodal coordinate direction*. The amount of the displacement is unknown until the analysis is completed. A set of coupled nodes which are not coincident, or which are not along the line of the coupled displacement direction, may produce an applied moment which will not appear

in the reaction forces. The actual degrees of freedom available for a particular node depends upon the degrees of freedom associated with element types [**ET**] at that node. For example, degrees of freedom available with BEAM3 elements are UX, UY, and ROTZ only. For scalar field analysis, this command is used to couple nodal temperatures, pressures, voltages, etc.

For an explicit dynamic analysis, the only valid DOF labels for coupling are UX, UY, and UZ. Since the rotational DOF (ROTX, ROTY, ROTZ) are not allowed, The **CP** family of commands should not be used in an explicit analysis to model rigid body behavior that involves rotations. If **CP** is used in this manner, it could lead to nonphysical responses.

A set of coupled nodes which are not coincident, or which are not along the line of the coupled displacement direction, produce an artificial moment constraint. If the structure rotates, a moment may be produced in the coupled set in the form of a force couple. This moment is in addition to the real reaction forces and may make it appear that moment equilibrium is not satisfied by just the applied forces and the reaction forces. Note, however, that in an explicit dynamic analysis, this artificial moment will not be produced. Rather, just the applied forces and the reaction forces will satisfy the moment equilibrium in the model. Thus, in an explicit analysis, the magnitude of nodal displacements for this set of nodes will depend on the distance from each node to the center of the coupled set, and the direction of displacement will depend on the resulting moment. This may lead to a nonphysical response in some cases.

Additional sets of coupled nodes may be generated from a specified set. Degrees of freedom are coupled within a set but are not coupled between sets. No degree of freedom should appear in more than one coupled set. Such an appearance would indicate that at least two sets were in fact part of a single larger set. The first degree of freedom of the coupled set is the "prime" degree of freedom. All other degrees of freedom in the coupled sets are eliminated from the solution matrices by their relationship to the prime degree of freedom. Forces applied to coupled nodes (in the coupled degree of freedom direction) will be summed and applied to the prime degree of freedom. Output forces are also summed at the prime degree of freedom. Degrees of freedom with specified constraints [**D**] should not be included in a coupled set (unless the degree of freedom is prime).

If master degrees of freedom are defined for coupled nodes, only the prime degree of freedom should be so defined. The use of coupled nodes reduces the set of coupled degrees of freedom to only one degree of freedom. The wavefront is correspondingly decreased; however, the overall stiffness (or conductivity) matrix formulation time is increased.

In FLOTRAN, you can apply periodic boundary conditions using the **CP** command along with the **PERI** macro. Attempts to use the **CP** command outside the context of the **PERI** macro may lead to unexpected results.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Couple DOFs
Main Menu>Preprocessor>Coupling / Ceqn>Cupl DOFs w/Mstr

CPCYC, *Lab*, *TOLER*, *KCN*, *DX*, *DY*, *DZ*, *KNONROT*

Couples the two side faces of a cyclically symmetric model for loadings that are the same on every segment.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

Lab

Degree of freedom label for coupled nodes (in the nodal coordinate system). If ALL, use all appropriate labels. Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations, in radians).

TOLER
Tolerance for coincidence (based on maximum coordinate difference in each global Cartesian direction for node locations and on angle differences for node orientations). Defaults to 0.0001. Only nodes within the tolerance are considered to be coincident for coupling.

KCN
In coordinate system *KCN*, node 1 of CP + dx dy dz = node 2 of CP.

DX, *DY*, *DZ*
Node location increments in the active coordinate system (DR, Dθ, DZ for cylindrical, DR, D θ, DΦ for spherical or toroidal).

KNONROT
When *KNONROT* = 0, the nodes on coupled sets are rotated into coordinate system *KCN* (see **NROTAT** command description). When *KNONROT* = 1, the nodes are not rotated, and you should make sure that coupled nodal DOF directions are correct.

Notes

Cyclic coupling requires identical node and element patterns on the low and high sector boundaries. The MSHCOPY operation allows convenient generation of identical node and element patterns. See Using CPCYC and MSHCOPY Commands in the *ANSYS Modeling and Meshing Guide* for more information.

Although developed primarily for use with cyclically symmetric models, your use of the **CPCYC** command is not limited to cyclic symmetry analyses.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Offset Nodes

CPDELE, *NSET1*, *NSET2*, *NINC*, *Nsel*

Deletes coupled degree of freedom sets.

PREP7: Coupled DOF

MP ME ST DY <> PR EM <> FL PP ED

NSET1, *NSET2*, *NINC*
Delete coupled sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL, *NSET2* and *NINC* are ignored and all coupled sets are deleted.

Nsel
Additional node selection control:

ANY
Delete coupled set if *any* of the selected nodes are in the set (default).

ALL
Delete coupled set only if *all* of the selected nodes are in the set.

Notes

See the **CP** command for a method to delete individual nodes from a set.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Del Coupled Sets
Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements

---

CPINTF, *Lab*, *TOLER*

Defines coupled degrees of freedom at an interface.

PREP7: Coupled DOF

MP ME ST DY <> PR EM <> FL PP ED

Lab

Degree of freedom label for coupled nodes (in the nodal coordinate system). If ALL, use all appropriate labels. Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations, in radians). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Fluid labels: PRES (pressure); VX, VY, or VZ (velocities). Electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials); CURR (current).

TOLER

Tolerance for coincidence (based on maximum coordinate difference in each global Cartesian direction for node locations and on angle differences for node orientations). Defaults to 0.0001. Only nodes within the tolerance are considered to be coincident for coupling.

Notes

Defines coupled degrees of freedom between coincident nodes (within a tolerance). May be used, for example, to "button" together elements interfacing at a seam, where the seam consists of a series of node pairs. One coupled set is generated for each selected degree of freedom for each pair of coincident nodes. For more than two coincident nodes in a cluster, a coupled set is generated from the lowest numbered node to each of the other nodes in the cluster. Coupled sets are generated only within (and not between) clusters. If fewer than all nodes are to be checked for coincidence, use the **NSEL** command to select nodes. Coupled set reference numbers are incremented by one from the highest previous set number. Use **CPLIST** to display the generated sets. Only nodes having the same nodal coordinate system orientations ("coincident" within a tolerance) are included. Use the **CEINTF** command to connect nodes by constraint equations instead of by coupling. Use the **EINTF** command to connect nodes by line elements instead of by coupling.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Coincident Nodes

/CPLANE, *KEY*

Specifies the cutting plane for section and capped displays.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

KEY

Specifies the cutting plane:

0
Cutting plane is normal to the viewing vector [**/VIEW**] and passes through the focus point [**/FOCUS**] (default).

1
The working plane [**WPLANE**] is the cutting plane.

Command Default

The cutting plane is normal to the viewing vector at the focus point.

Notes

Defines the cutting plane to be used for section and capped displays [**/TYPE,,**(1, 5, or 7)].

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

CPLGEN, *NSETF*, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*

Generates sets of coupled nodes from an existing set.

PREP7: Coupled DOF

MP ME ST DY <> PR EM <> FL PP ED

NSETF
Generate sets from existing set *NSETF*.

Lab1, *Lab2*, *Lab3*, *Lab4*, *Lab5*
Generate sets with these labels (see **CP** command for valid labels). Sets are numbered as the highest existing set number + 1.

Notes

Generates additional sets of coupled nodes (with different labels) from an existing set [**CP**, **CPNGEN**]. The same node numbers are included in the generated sets. If all labels of nodes are to be coupled and the nodes are coincident, the **NUMMRG** command should be used to automatically redefine the node number (for efficiency).

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Gen w/Same Nodes

CPLIST, *NSET1*, *NSET2*, *NINC*, *Nsel*

Lists the coupled degree of freedom sets.

PREP7: Coupled DOF

MP ME ST DY <> PR EM <> FL PP ED

NSET1, *NSET2*, *NINC*

List coupled sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL (default), *NSET2* and *NINC* are ignored and all coupled sets are listed.

Nsel

Node selection control:

ANY

List coupled set if *any* of the selected nodes are in the set (default).

ALL

List coupled set only if *all* of the selected nodes are in the set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Other>Coupled Sets>All CP nodes selected
Utility Menu>List>Other>Coupled Sets>Any CP node selected

CPMERGE, *Lab*

Merges different couple sets with duplicate degrees of freedom into one couple set.

PREP7: Coupled DOF

MP ME ST DY <> PR EM <> FL PP ED

Lab

Degree of freedom label for coupled nodes (in the nodal coordinate system). Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations) (in radians). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Fluid labels: PRES (pressure); VX, VY, or VZ (velocities). Electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials); CURR (current). Explicit analysis labels: UX, UY, or UZ (displacements). The degree of freedom set is determined from all element types defined and the **DOF** command, if used.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Capacitor
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Curr Cntl CS
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Curr Cntl VS
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Diode
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Constant Amplitude

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Exponential
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Piecewise
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Pulse
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Sinusoidal
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Constant Amplitude
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Exponential
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Piecewise
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Pulse
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Sinusoidal
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Inductor
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Mass Cond 2D
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Mass Cond 3D
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Mutual Ind
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Resistor
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Strnd Coil
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Vltg Cntl CS
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Vltg Cntl VS
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Wire
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Zener Diode
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Linear Rotary
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Linear Trans
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Nonlin Rotary
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Nonlin Trans
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Slide Film
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Mass
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Linear Rotary
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Linear Trans
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Rotary
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Trans
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Capacitor
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Constant Amplitude
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Exponential
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Piecewise
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Pulse
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Sinusoidal
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Constant Amplitude
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Exponential
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Piecewise
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Pulse
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Sinusoidal
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Inductor
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Resistor
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Wire
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Transducer>ElecMech
Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements

CPNGEN, *NSET*, *Lab*, *NODE1*, *NODE2*, *NINC*

Defines, modifies, or adds to a set of coupled degrees of freedom.

PREP7: Coupled DOF

MP ME ST DY <> PR EM <> FL PP ED

NSET
Set reference number [**CP**].

Lab
Degree of freedom label [**CP**].

NODE1, *NODE2*, *NINC*
Include in coupled set nodes *NODE1* to *NODE2* in steps of *NINC* (defaults to 1). If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If -*NODE1*, delete range of nodes from set instead of including. A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Notes

Defines, modifies, or adds to a set of coupled degrees of freedom. May be used in combination with (or in place of) the **CP** command. Repeat **CPNGEN** command for additional nodes.

Menu Paths

This command cannot be accessed from a menu.

CPSGEN, *ITIME*, *INC*, *NSET1*, *NSET2*, *NINC*

Generates sets of coupled nodes from existing sets.

PREP7: Coupled DOF

MP ME ST DY <> PR EM <> FL PP ED

ITIME, *INC*
Do this generation operation a total of *ITIME*s, incrementing all nodes in the existing sets by *INC* each time after the first. *ITIME* must be > 1 for generation to occur.

NSET1, *NSET2*, *NINC*
Generate sets from sets beginning with *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* is negative, *NSET2* and *NINC* are ignored and the last |*NSET1*| sets (in sequence from the maximum set number) are used as the sets to be repeated.

Notes

Generates additional sets of coupled nodes (with the same labels) from existing sets. Node numbers between sets may be uniformly incremented.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Gen w/Same DOF

CQC, *SIGNIF*, *Label*

Specifies the complete quadratic mode combination method.

SOLUTION: Spectrum Options

MP ME ST <> <> PR <> <> <> PP ED

SIGNIF

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single point, multi-point, or DDAM response (**SPOPT**,SPRS, MPRS or DDAM), the significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT**,PSD.)

Label

Label identifying the combined mode solution output.

DISP

Displacement solution (default). Displacements, stresses, forces, etc., are available.

VELO

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

ACEL

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

Notes

Damping is required for this mode combination method. The **CQC** command is also valid for PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine

---

CRPLIM, *CRCR*, *Option*

Specifies the creep criterion for automatic time stepping.

SOLUTION: Nonlinear Options

MP ME ST <> <> <> <> <> <> PP ED

CRCR

Value of creep criteria for the creep limit ratio control.

Option

Type of creep analysis:

1 or ON
Implicit creep analysis.

0 or OFF
Explicit creep analysis.

Notes

The creep ratio control can be used at the same time for implicit creep and explicit creep analyses. For implicit creep (*Option* = 1), the default value of *CRCR* is zero (i.e., no creep limit control), and you are allowed to specify any value. For explicit creep (*Option* = 0), the default value of *CRCR* is 0.1. The maximum value allowed is 0.25. This command is also valid in PREP7. The **CUTCONTROL** command can be used for the same purpose and is the preferred command, when **SOLCONTROL** is ON.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Creep Criterion
Main Menu>Solution>Load Step Opts>Nonlinear>Creep Criterion

CS, *KCN*, *KCS*, *NORIG*, *NXAX*, *NXYPL*, *PAR1*, *PAR2*

Defines a local coordinate system by three node locations.

DATABASE: Coordinate System

MP ME ST DY <> PR EM <> FL PP ED

KCN
Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

KCS
Coordinate system type:

0 or CART
Cartesian

1 or CYLIN
Cylindrical (circular or elliptical)

2 or SPHE
Spherical (or spheroidal)

3 or TORO
Toroidal

NORIG
Node defining the origin of this coordinate system. If *NORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NXAX
Node defining the positive x-axis orientation of this coordinate system.

NXYPL
Node defining the x-y plane (with *NORIG* and *NXAX*) in the first or second quadrant of this coordinate system.

PAR1
Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

PAR2
Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

Notes

Defines and activates a local right-handed coordinate system by specifying three existing nodes: to locate the origin, to locate the positive x-axis, and to define the positive x-y plane. This local system becomes the active coordinate system. See the **CLOCAL**, **CSKP**, **CSWPLA**, and **LOCAL** commands for alternate definitions. Local coordinate systems may be displayed with the **/PSYMB** command.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>By 3 Nodes

---

CSCIR, *KCN*, *KTHET*, *KPHI*

Locates the singularity for non-Cartesian local coordinate systems.

DATABASE: Coordinate System

MP ME ST DY <> PR EM <> FL PP ED

KCN
Number of the local coordinate system in which singularity location is to be changed. Must be greater than 10.

KTHET
Theta singularity location for cylindrical, spherical, and toroidal systems:

0
Singularity at ±180°.

1
Singularity at 0° (360°).

KPHI
Phi singularity location for toroidal systems:

0
Singularity in phi direction at ±180°.

1
Singularity in phi direction at 0° (360°).

Command Default

Singularities at ±180°.

Notes

Continuous closed surfaces (circles, cylinders, spheres, etc.) have a singularity (discontinuity) at $\theta = \pm 180°$. For local cylindrical, spherical, and toroidal coordinate systems, this singularity location may be changed to 0° (360°).

An additional, similar singularity occurs in the toroidal coordinate system at $\Phi = \pm 180°$ and can be moved with *KPHI*. Additional singularities occur in the spherical coordinate system at $\Phi = \pm 90°$, but cannot be moved.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Local Coordinate Systems>Move Singularity

---

CSDELE, *KCN1*, *KCN2*, *KCINC*

Deletes local coordinate systems.

DATABASE: Coordinate System

MP ME ST DY <> PR EM <> FL PP ED

KCN1, *KCN2*, *KCINC*

Delete coordinate systems from *KCN1* (must be greater than 10) to *KCN2* (defaults to *KCN1*) in steps of *KCINC* (defaults to 1). If *KCN1* = ALL, *KCN2* and *KCINC* are ignored and all coordinate systems are deleted.

Notes

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node
Utility Menu>WorkPlane>Local Coordinate Systems>Delete Local CS

CSKP, *KCN*, *KCS*, *PORIG*, *PXAXS*, *PXYPL*, *PAR1*, *PAR2*

Defines a local coordinate system by three keypoint locations.

DATABASE: Coordinate System

MP ME ST DY <> PR EM <> FL PP ED

KCN

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

KCS

Coordinate system type:

0 or CART
Cartesian

1 or CYLIN
Cylindrical (circular or elliptical)

2 or SPHE
Spherical (or spheroidal)

3 or TORO
Toroidal

PORIG
Keypoint defining the origin of this coordinate system. If *PORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

PXAXS
Keypoint defining the positive x-axis orientation of this coordinate system.

PXYPL
Keypoint defining the x-y plane (with *PORIG* and *PXAXS*) in the first or second quadrant of this coordinate system.

PAR1
Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

PAR2
Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

Notes

Defines and activates a local right-handed coordinate system by specifying three existing keypoints: to locate the origin, to locate the positive x-axis, and to define the positive x-y plane. This local system becomes the active coordinate system. See the **CLOCAL**, **CS**, **CSWPLA**, and **LOCAL** commands for alternate definitions. Local coordinate systems may be displayed with the **/PSYMB** command.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>By 3 Keypoints

---

CSLIST, *KCN1*, *KCN2*, *KCINC*

Lists coordinate systems.

DATABASE: Coordinate System

MP ME ST DY <> PR EM <> FL PP ED

KCN1, *KCN2*, *KCINC*
List coordinate systems from *KCN1* to *KCN2* (defaults to *KCN1*) in steps of *KCINC* (defaults to 1). If *KCN1* = ALL (default), *KCN2* and *KCINC* are ignored and all coordinate systems are listed.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Other>Local Coord Sys

CSWPLA, *KCN*, *KCS*, *PAR1*, *PAR2*

Defines a local coordinate system at the origin of the working plane.

DATABASE: Coordinate System

MP ME ST DY <> PR EM <> FL PP ED

KCN

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

KCS

Coordinate system type:

0 or CART
: Cartesian

1 or CYLIN
: Cylindrical (circular or elliptical)

2 or SPHE
: Spherical (or spheroidal)

3 or TORO
: Toroidal

PAR1

Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

PAR2

Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

Notes

Defines and activates a local right-handed coordinate system centered at the origin of the working plane. The coordinate system's local x-y plane (for a Cartesian system) or R-θ plane (for a cylindrical or spherical system) corresponds to the working plane. This local system becomes the active coordinate system. See the **CS**, **LOCAL**, **CLOCAL**, and **CSKP** commands for alternate ways to define a local coordinate system. Local coordinate systems may be displayed with the **/PSYMB** command.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane
Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane
Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>At WP Origin

---

CSYS, *KCN*

Activates a previously defined coordinate system.

DATABASE: Coordinate System

MP ME ST DY <> PR EM <> FL PP ED

KCN

Specifies the active coordinate system, as follows:

0 (default)
: Cartesian

1
: Cylindrical with Z as the axis of rotation

2
: Spherical

4 or WP
: Working Plane

5
: Cylindrical with Y as the axis of rotation

11 or greater
: Any previously defined local coordinate system

Notes

The **CSYS** command activates a previously defined coordinate system for geometry input and generation. The **LOCAL**, **CLOCAL**, **CS**, **CSKP**, and **CSWPLA** commands also activate coordinate systems as they are defined.

The active coordinate system for files created via the **CDWRITE** command is Cartesian (**CSYS**,0).

This command is valid in any processor.

CSYS,4 (or **CSYS**,WP) activates working plane tracking, which updates the coordinate system to follow working plane changes. To deactivate working plane tracking, activate *any other* coordinate system (for example, **CSYS**,0 or **CSYS**,11).

CSYS,5 is a cylindrical coordinate system with Y as the axis. The X, Y and Z axes are radial, theta, and axial (respectively). However, **CSYS**,5 has a specific orientation: the local X is in the global +X direction, local Y is in the global -Z direction, and local Z (the cylindrical axis) is in the global +Y direction. The R-Theta plane is then the global X-Z plane, as it is for an axisymmetric model.

Menu Paths

Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node
Utility Menu>WorkPlane>Change Active CS to>Global Cartesian
Utility Menu>WorkPlane>Change Active CS to>Global Cylindrical
Utility Menu>WorkPlane>Change Active CS to>Global Spherical
Utility Menu>WorkPlane>Change Active CS to>Specified Coord Sys
Utility Menu>WorkPlane>Change Active CS to>Working Plane
Utility Menu>WorkPlane>Offset WP to>Global Origin

/CTYPE, *KEY*, *DOTD*, *DOTS*, *DSHP*, *TLEN*

Specifies the type of contour display.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

KEY
Type of display:

0
Standard contour display.

1
Isosurface display.

2
Particle gradient display.

3
Gradient triad display.

DOTD
Maximum dot density for particle gradient display (*KEY* = 2). Density is expressed as dots per screen width (defaults to 30).

DOTS
Dot size for particle gradient display (*KEY* = 2). Size is expressed as a fraction of the screen width (defaults to 0.0 (single dot width)).

DSHP
Spherical dot shape precision for particle gradient display (*KEY* = 2). (3-D options are supported only on 3-D devices):

0
Flat 2-D circular dot.

1
Flat-sided 3-D polyhedron.

n
3-D sphere with *n* (>1) polygon divisions per 90° of radius.

TLEN
Maximum length of triads for gradient triad display (*KEY* = 3). Value is expressed as a fraction of the screen width (defaults to 0.067).

Command Default

Standard contour display.

Notes

Use **/CTYPE**,STAT to display the current settings. Only the standard contour display [**/CTYPE**,0) and the isosurface contour display [**/CTYPE**,1] are supported by PowerGraphics [**/GRAPHICS**,POWER].

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Contour Style

CURR2D

Calculates current flow in a 2-D conductor.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

Notes

CURR2D invokes an ANSYS macro which calculates the total current flowing in a conducting body for a 2-D planar or axisymmetric magnetic field analysis. The currents may be applied source currents or induced currents (eddy currents). The elements of the conducting region must be selected before this command is issued. The total current calculated by the macro is stored in the parameter TCURR. Also, the total current and total current density are stored on a per-element basis in the element table [**ETABLE**] with the labels TCURR and JT, respectively. Use the **PLETAB** and **PRETAB** commands to plot and list the element table items.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Element Based>Current

---

CUTCONTROL, *Lab*, *VALUE*, *Option*

Controls time-step cutback during a nonlinear solution.

SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> <> PP ED

Lab

Specifies the criteria for causing a cutback. Possible arguments are

PLSLIMIT

Maximum equivalent plastic strain allowed within a time-step (substep). If the calculated value exceeds the *VALUE*, ANSYS performs a cutback (bisection). *VALUE* defaults to 0.15 (15%).

CRPLIMIT

Maximum equivalent creep ratio allowed within a time step (substep). If the calculated value exceeds *VALUE*, ANSYS performs a cutback (bisection). The default depends on whether you are performing an implicit or explicit creep analysis (see *Option* argument below). The creep ratio control can be used at the same time for implicit creep and explicit creep analyses. For an implicit creep analysis, *VALUE* defaults to zero (i.e., no creep limit control), and you are allowed to enter any value. For an explicit creep analysis, *VALUE* defaults to 0.1 (10%) and the maximum value allowed is 0.25.

DSPLIMIT

Maximum incremental displacement within the solution field in a time step (substep). If the maximum calculated value exceeds *VALUE*, ANSYS performs a cutback (bisection). *VALUE* defaults to 1.0×10^{7}.

NPOINT
Number of points in a cycle for a second order dynamic equation, used to control automatic time stepping. If the number of solution points per cycle exceeds *VALUE*, ANSYS performs a cutback in time step size. *VALUE* defaults to 13.

NOITERPREDICT
If *VALUE* is 0 (default), an internal auto time step scheme will predict the number of iterations for nonlinear convergence and perform a cutback earlier than the number of iterations specified by the **NEQIT** command. This is the recommended option. If *VALUE* is 1, the solution will iterate (if nonconvergent) to **NEQIT** number of iterations before a cutback is invoked. It is sometimes useful for poorly-convergent problems, but rarely needed in general.

VALUE
Numeric value for the specified cutback criterion. For *Lab* = CRPLIMIT, *VALUE* is the creep criteria for the creep limit ratio control.

Option
Type of creep analysis. *Option* is valid for *Lab* = CRPLIMIT only.

1 or ON
Implicit creep analysis.

0 or OFF
Explicit creep analysis.

Notes

The default values given for this command assume **SOLCONTROL**,ON (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL**,ON and **SOLCONTROL**,OFF.

A cutback is a method for automatically reducing the step size when either the solution error is too large or the solution encounters convergence difficulties during a nonlinear analysis. Should a convergence failure occur, ANSYS will reduce the time step interval to a fraction of its previous size and automatically continue the solution from the last successfully converged time step. If the reduced time step again fails to converge, ANSYS will again reduce the time step size and proceed with the solution. This process continues until convergence is achieved or the minimum specified time step value is reached.

The **CRPLIM** command is functionally equivalent to *Lab* = CRPLIMIT.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Cutback Control
Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Solution>Load Step Opts>Nonlinear>Cutback Control

/CVAL, *WN*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*, *V7*, *V8*

Specifies nonuniform contour values on stress displays.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

V1, *V2*, *V3*, *V4*, *V5*, *V6*, *V7*, *V8*
Up to 8 contour values may be specified (in ascending order). The 0.0 value (if any) must not be the last value specified. If no values are specified, all contour specifications are erased and contours are automatically calculated.

Command Default

Nine contour values uniformly spaced between the extreme values.

Notes

This command is similar to the **/CONTOUR** command. With **/CVAL**, however, you define the upper level of each contour band instead of having the contours uniformly distributed over the range. The minimum value (including a zero value for the first band) for a contour band cannot be specified. If you use both **/CONTOUR** and **/CVAL**, the last command issued takes precedence.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Non-uniform Contours

CVAR, *IR*, *IA*, *IB*, *ITYPE*, *DATUM*, *Name*

Computes covariance between two quantities.

POST26: Special Purpose

MP ME ST <> <> <> <> <> <> PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previous variable, the previous variable will be overwritten with this result.

IA, *IB*
Reference numbers of the two variables to be operated on. If only one, leave *IB* blank.

ITYPE
Defines the type of response PSD to be calculated:

0,1
Displacement (default).

2
Velocity.

3
Acceleration.

DATUM
Defines the reference with respect to which covariance is to be calculated:

1
Absolute value.

2
Relative to base (default).

Name
Thirty-two character name for identifying the variable on listings and displays. Embedded blanks are compressed upon output.

Notes

This command computes the covariance value for the variables referenced by the reference numbers *IA* and *IB*. If *DATUM* = 2, the variable referenced by *IR* will contain the individual modal contributions (i.e., the dynamic or relative values). If *DATUM* = 1, the variable referenced by *IR* will contain the modal contributions followed by the contributions of pseudo-static and covariance between dynamic and pseudo-static responses. **File.PSD** must be available for the calculations to occur.

Menu Paths

Main Menu>TimeHist Postpro>Calc Covariance

/CWD, *DIRPATH*
Changes the current working directory.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

DIRPATH
The full path name of the new working directory.

Notes

After issuing the /CWD command, all new files opened with no default directory specified (via the **FILE**, **/COPY**, or **RESUME** commands, for example) default to the new *DIRPATH* directory.

If issuing the command interactively and the specified directory does not exist, no change of directory occurs and the command generates an error message. If the command executes via a batch run and the specified directory does not exist, the batch program terminates with an error.

Menu Paths

Utility Menu>File>Change Directory

/CYCEXPAND, *WN*, *OPTION*, *Value1*, *Value2*

Graphically expands displacements, stresses and strains of a cyclically symmetric model.

PREP7: Special Purpose
POST1: Special Purpose

MP ME ST <> <> PR <> <> <> PP ED

WN

The window number to which the expansion applies. Valid values are 1 through 5. The default value is 1. The window number applies only to the AMOUNT argument.

OPTION

One of the following options:

ON

Activates cyclic expansion using the previous settings (if any). If no previous settings exist, this option activates the default settings.

DEFAULT

Resets cyclic expansion to the default settings.

OFF

Deactivates cyclic expansion. This option is the default.

STATUS

Lists the current cyclic expansion settings.

AMOUNT

The number of repetitions *or* the total angle.

Value1

NREPEAT

Value2

The number of repetitions. The default is the total number of sectors in 360 degrees.

or

Value1

ANGLE

Value2

The total angle in degrees. The default is 360.

WHAT

A specified portion or subset of the model to expand:

Value1

The component name of the elements to expand. The default is all selected components.

EDGE

Sector edge display key.

1

Suppresses display of edges between sectors even if the cyclic count varies between active windows.

Caution: Plots with fewer than the maximum number of repetitions may have missing element faces at the sector boundaries.

0 or OFF
Averages stresses or strains across sector boundaries. This value is the default (although the default reverts to 1 or ON if the cyclic count varies between active windows).

1 or ON
No averaging of stresses or strains occurs and sector boundaries are shown on the plot.

PHASEANG
The phase angle shift:

Value1
The phase angle shift in degrees. The valid range is 0 through 360. The default is 0. Typically, this value is the phase angle obtained via the **CYCPHASE** command.

Command Default

The default **/CYCEXPAND** command (issuing the command with no arguments) deactivates cyclic expansion (*OPTION* = OFF). The default window number (*WN*) is 1.

Notes

In preprocessing, the **/CYCEXPAND** command verifies a cyclically symmetric model by graphically expanding it partially or through the full 360 degrees.

For the postprocessing plot nodal solution (**PLNSOL**) operation, the command graphically expands displacements, stresses and strains of a cyclically symmetric model partially or though the full 360 degrees by combining the *real* (original nodes and elements) and *imaginary* (duplicate nodes and elements) parts of the solution.

For the print nodal solution (**PRNSOL**) operation, the command expands the printed output of displacements or stresses on a sector-by-sector basis.

Use of the **/CYCEXPAND** command does *not* change the database. The command does not modify the geometry, nodal displacements or element stresses.

The command affects element and result plots only. It has no effect on operations other than plot element solution (**PLESOL**), plot nodal solution (**PLNSOL**) and print nodal solution (**PRNSOL**). Operations other than **PLESOL**, **PLNSOL**, or **PRNSOL** work on the *unprocessed* real and imaginary parts of a cyclic symmetry solution

If you issue a **/CYCEXPAND**,,OFF command, you cannot then expand the model by simply issuing another **/CYCEXPAND** command (for example, to specify an NREPEAT value for the number of repetitions). In such a case, you must specify **/CYCEXPAND**,,ON, which activates expansion using the previous settings (if any) or the default settings.

The command requires PowerGraphics. (By default, PowerGraphics is active during an interactive session, but not during a batch run.) To activate PowerGraphics, issue the **/GRAPHICS**,POWER command. Any setting which bypasses PowerGraphics (for example, **RSYS**,SOLU) also bypasses cyclic expansion; in such cases, the **/CYCEXPAND** command displays unprocessed real and imaginary results.

The **/CYCEXPAND** command does not work with PGR files.

The **/CYCEXPAND** command is incompatible with the **/ESHAPE**,1 or **/ESHAPE**,*FAC* command.

The **/PBC** (plot boundary conditions) command is deactivated when cyclic expansion is active (**/CYCEXPAND**,,ON). To view BCs on the basic sector, deactivate cyclic expansion (**/CYCEXPAND**,,OFF) and issue this command: **/PBC**,ALL,,1

In cases involving non-cyclically symmetric loading, all boundary conditions (BCs) will plot when **/CYCEXPAND** is active. When examining the plot, however, it is not possible to distinguish between BCs that exist on every sector and those that exist (via a table referencing SECTOR) only on some sectors or which vary in value. (Only the basic sector is plotted.)

For magnetic cyclic symmetry analyses, the **/CYCEXPAND** command produces contour plots but not vector plots.

To learn more about analyzing a cyclically symmetric structure, see Cyclic Symmetry Analysis in the *ANSYS Advanced Analysis Techniques Guide*.

Menu Paths

Main Menu>General Postproc>Cyclic Analysis>Cyc Expansion
Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyc Expansion

CYCLIC, *NSECTOR*, *ANGLE*, *KCN*, *Name*, *USRCOMP*

Specifies a cyclic symmetry analysis.

PREP7: Special Purpose

MP ME ST <> <> PR <> <> <> PP ED

NSECTOR
The number of sectors in the full 360 degrees, *or* one of the following options:

STATUS
Indicates the current cyclic status.

OFF
Resets model to normal (non-cyclic) status and removes the duplicate sector if it exists. This option also deletes automatically detected edge components (generated when *USRCOMP* = 0).

UNDOUBLE
Removes the duplicate sector if it exists. The duplicate sector is created during the solution (**SOLVE**) stage of a modal cyclic symmetry analysis.

Note — The duplicate sector is necessary for displaying cyclic symmetry analysis results during postprocessing (**/POST1**).

If you specify a value of STATUS, OFF or UNDOUBLE, the command ignores all remaining arguments.

ANGLE
The sector angle in degrees.

KCN
An arbitrary reference number assigned to the cyclic coordinate system. The default value of 0 specifies automatic detection.

Name

The root name of sector low- and high-edge components (line, area, or node components). The default root name (when *USRCOMP* = 0) is "CYCLIC". A root name that you specify can contain up to 11 characters.

The naming convention for each low- and high-edge component pair is either of the following:

*name*_m*xx*l, *name*_m*xx*h (potentially matched node patterns)
*name*_u*xx*l, *name*_u*xx*h (potentially unmatched node patterns)

The *name* value is the default ("CYCLIC") or specified root name and *xx* is the component pair ID number (sequential, starting at 01).

USRCOMP

The number of pairs of user-defined low- and high-edge components on the cyclic sector (if any). The default value of 0 specifies automatic detection of sector edges; however, the automatic setting is not valid in all cases. (For more information, see the Notes section below.) If the value is greater than 0, no verification of user-defined components occurs.

Command Default

The default **CYCLIC** command (issuing the command with no arguments) detects the number of sectors (*NSECTOR*), the sector angle (*ANGLE*), and the coordinate system (*KCN*) based upon the existing solid or finite-element model. The command also detects sector low- and high-edge components in most cases and assigns the default root name "CYCLIC" to the components.

Notes

You can input your own value for *NSECTOR*, *ANGLE* or *KCN*; if you do so, the command verifies argument values before executing.

When *USRCOMP* = 0 (default), the **CYCLIC** command automatically detects low- and high-edge components for models comprised of any combination of line, area, or volume elements. If a solid model exists, however, the command uses only the lines, areas, and/or volumes to determine the low- and high-edge components; the elements, if any, are ignored.

If you issue a **CYCOPT**,TOLER command to set a tolerance for edge-component pairing *before* issuing the **CYCLIC** command, the **CYCLIC** command uses the specified tolerance when performing automatic edge-component detection.

For 2-D models, autodetection does not consider the **CSYS**,5 or **CSYS**,6 coordinate system specification. Auto-detection for 180 degree (two-sector) models is not possible unless a central hole exists.

The **CYCLIC** command sets values and keys so that, if possible, the area-mesh (**AMESH**) or volume-mesh (**VMESH**) command meshes the sector with matching node and element face patterns on the low and high edges. (The command has no effect on any other element-creation command.)

Issue the **CYCLIC** command *prior* to the meshing command to, if possible, produce a mesh with identical node and element patterns on the low and high sector edges. Only the **AMESH** or **VMESH** commands can perform automated matching. (Other meshing operation commands such as **VSWEEP** cannot.) If you employ a meshing operation other than **AMESH** or **VMESH**, you should ensure that node and element face patterns match, if desired. The **CYCLIC** command output indicates whether each edge-component pair has or can produce a matching node pair.

A cyclic solution (via the **SOLVE** command) allows dissimilar mesh patterns on the extreme boundaries of a cyclically symmetric model. The allowance for dissimilar patterns is useful when you have only finite-element meshes for your model but not the geometry data necessary to remesh it to obtain identical node patterns. In such cases, it is possible to obtain solution results, although perhaps at the expense of accuracy. A warning message appears because results may be degraded near the sector edges.

The constraint equations (CEs) that tie together the low and high edges of your model are generated at the solution stage of the analysis from the low- and high-edge components (and nowhere else). You should verify that automatically detected components are in the correct locations and that you are able to account for all components; to do so, you can list (**CMLIST**) or plot (**CMPLOT**) the components.

To learn more about analyzing a cyclically symmetric structure, see Cyclic Symmetry Analysis in the *ANSYS Advanced Analysis Techniques Guide*.

Menu Paths

Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Model>Auto Defined
Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Model>Status
Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Model>User Defined
Main Menu>Preprocessor>Modeling>Cyclic Sector>Del Dupl Sector
Main Menu>Preprocessor>Modeling>Cyclic Sector>Reset (OFF)

CYCOPT, *OPTION*, *Value1*, *Value2*, *Value3*, *Value4*, *Value5*, *Value6*, *Value7*

Specifies solution options for a cyclic symmetry analysis.

PREP7: Special Purpose
SOLUTION: Dynamic Options

MP ME ST <> <> PR <> <> <> PP ED

OPTION
One of the following options:

STATUS
Lists the solution option settings active for the cyclic model.

DEFAULT
Sets the default cyclic solution settings.

HINDEX
The harmonic index solution ranges for modal or buckling cyclic symmetry analyses. The **SOLVE** command initiates a cyclic symmetry solution sequence at the harmonic indices specified. (By default, the **SOLVE** command solves for *all* available harmonic indices.) Static cyclic symmetry solutions always use all harmonic indices required for the applied loads.

EVEN / ODD
For low-frequency electromagnetic analysis only, EVEN specifies a symmetric solution and ODD specifies an antisymmetric solution.

The value you specify is based on the harmonic index: EVEN (default) indicates harmonic index = 0, and ODD indicates harmonic index = *N* / 2 (where *N* is an integer representing the number of sectors in 360°). A value of ODD applies only when *N* is an even number.

The **CYCOPT** command with this HINDEX option is cumulative. To remove an option (for example, EVEN), issue this command: **CYCOPT**,HINDEX,EVEN,,,-1

Value1, *Value2*, *Value3*
Solve harmonic indices in range *Value1* through *Value2* in steps of *Value3*. Repeat the command to add other ranges. The default solves *all* applicable harmonic indices.

Value4
The only valid value is -1. If specified, it removes *Value1* through *Value2* in steps of *Value3* from the set to solve.

LDSECT
Restricts subsequently defined DOF constraint values, force loads, and surface loads to a specified sector. The restriction remains in effect until you change or reset it.

Value1
The sector number. A value other than 0 (default) is valid for a cyclic symmetry analysis with non-cyclically symmetric loading only. A value of 0 (or ALL) resets the default behavior for cyclic loading (where the loads are identical on all sectors).

COMBINE
For linear static cyclic symmetry analysis with non-cyclically symmetric loading only, expands and combines all harmonic index solutions and writes them to the results file during the solution phase of the analysis.

Value1
To enable combining of harmonic index solutions, specify YES, ON, or 1.

To disable combining (the default behavior), specify NO, OFF, or 0.

DOF
The degrees of freedom to couple from the nodes on the low sector boundary to nodes on the high boundary:

Value1
The component pair ID number.

Value2-Value7
The constraint-equation/-coupling degree of freedom (DOF) for this pair. Repeat the command to add other DOFs. The default is constraint-equation/-coupling *all* applicable DOFs.

TOLER
The tolerance used to determine whether a node on the low edge is paired with a node on the high edge.

Value1
The tolerance value, as follows:

> 0 -- The absolute distance tolerance for automatic sector-boundary detection and low-/high-edge component node pairing.

< 0 -- The relative tolerance for automatic sector-boundary detection and low-/high-edge component node pairing. In this case, the tolerance is *Value1* * *Length*, where *Length* is the length of the diagonal of an imaginary box enclosing the model.

The default tolerance is -1.0E-4 (also represented by *Value1* = 0).

MOVE
Flag specifying whether ANSYS should move high- or low-edge component nodes paired within the specified tolerance (TOLER) to create *precisely matching* pairs.

Value1
The flag value, as follows:

0 -- Do not move edge component nodes. This value is the default.

1 *or* HIGH -- Move the high-edge component nodes to precisely match the low-edge component nodes.

-1 *or* LOW -- Move the low-edge component nodes to precisely match the high-edge component nodes.

USRROT
Flag specifying whether ANSYS should override automatic nodal rotations to edge components and allow you to apply nodal rotations manually.

Value1
The flag value, as follows:

0 (OFF *or* NO) -- Allow automatic node rotation. This behavior is the default.

1 (ON *or* YES) -- Suppress automatic node rotation. If you select this option, you *must* apply appropriate nodal rotations to all edge component nodes; otherwise, your analysis will yield incorrect solution results.

LOW -- Suppresses automatic rotation of low-edge component nodes only, allowing you to apply them manually. (Automatic rotation of high-edge component nodes occurs to produce the matching edge nodes required for a valid cyclic solution.)

HIGH -- Suppresses automatic rotation of high-edge component nodes only, allowing you to apply them manually. (Automatic rotation of low-edge component nodes occurs to produce the matching edge nodes required for a valid cyclic solution.)

Command Default

No defaults are available for the **CYCOPT** command. You must specify an argument (*OPTION*) when issuing the command. Other values which may be necessary depend upon which argument you specify.

Notes

ANSYS solves a cyclically symmetric model (set up via the **CYCLIC** command during preprocessing) at the harmonic indices specified via the **CYCOPT** command.

The **CYCOPT**,COMBINE option is an alternative to the **/CYCEXPAND** command and is especially useful for testing purposes. However, ANSYS recommends specifying COMBINE only when the number of sectors is relatively small. (The option expands nodes and elements into the full 360° and can slow postprocessing significantly.

If you issue a **CYCOPT**,TOLER command to set a tolerance for edge-component pairing *before* issuing the **CYCLIC** command, the **CYCLIC** command uses the specified tolerance when performing automatic edge-component detection.

In cases involving non-cyclically symmetric loading (that is, when LDSECT > 0), the underlying command operations create or modify the required SECTOR tabular boundary condition (BC) data to apply on the appropriate sector. Therefore, it is not necessary to manipulate tables for situations where the applied BC is not a function of other tabular BC variables such as TIME, X, Y, Z, and so on.

To delete a previously applied load on a specified sector, issue an **FDELE** command.

Because edge nodes are rotated into the cyclic coordinate system during solution, any applied displacements or forces on sector edges will be in the cyclic coordinate system.

The **CYCOPT** command is valid in the preprocessing and solution stages of an analysis.

To learn more about analyzing a cyclically symmetric structure, see Cyclic Symmetry Analysis in the *ANSYS Advanced Analysis Techniques Guide*.

Menu Paths

Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Options
Main Menu>Solution>Solve>Cyclic Options

CYCPHASE, *TYPE*, *OPTION*

Provides tools for determining minimum and maximum possible result values from frequency couplets produced in a modal cyclic symmetry analysis.

POST1: Special Purpose

MP ME ST <> <> PR <> <> <> PP <>

TYPE
The type of operation requested:

DISP
Calculate the maximum and minimum possible displacement at each node in the original sector model. Store the values and the phase angle at which they occurred.

STRESS
Calculate the maximum and minimum possible stresses at each node in the original sector model. Store the values and the phase angle at which they occurred.

STRAIN
Calculate the maximum and minimum possible strains at each node in the original sector model. Store the values and the phase angle at which they occurred.

ALL
Calculate the maximum and minimum possible displacement, stress and strain at each node in the original sector model. Store the values and the phase angle at which they occurred.

GET
Places the value of a MAX or MIN item into the _CYCVALUE parameter, the node for that value in the _CYCNODE parameter, and the phase angle for the value in the _CYCPHASE parameter.

PUT
Put resulting sweep values for printing (via the **PRNSOL** command) or plotting (via the **PLNSOL** command).

LIST
List the current minimum/maximum displacement, stress and strain nodal values.

STAT
Summarize the results of the last phase sweep.

CLEAR
Clear phase-sweep information from the database.

OPTION
If *TYPE* = DISP, STRAIN, STRESS or ALL, controls the sweep angle increment to use in the search:

Angle
The sweep angle increment in degrees, greater than 0.1 and less than 10. The default is 1.

If *TYPE* = PUT, controls which values are placed onto the model:

MAX
Put all existing nodal maximum values onto the model. This option is the default.

MIN
Put all existing nodal minimum values onto the model.

If *TYPE* = GET, controls the values placed into cyclic parameters:

Item
Specifies the type of values on which to operate:

U -- Displacement
S -- Stress
EPEL -- Strain

Comp
Specifies the specific component of displacement, stress or strain for which to get information:

X,Y,Z -- Basic components
XY,YZ,XZ -- Shear components
1,2,3 -- Principal values
EQV -- Equivalent value
SUM -- USUM

MxMn
Specifies whether the requested value information is for the maximum or minimum value:

MAX -- Maximum value.
MIN -- Minimum value.

Command Default

No defaults are available for the **CYCPHASE** command. You must specify an argument (*TYPE*) when issuing the command. Other values which may be necessary (*OPTION*) depend upon which *TYPE* argument you specify.

Notes

When you expand the results of a modal cyclic symmetry analysis (via the **/CYCEXPAND** or **EXPAND** command), ANSYS combines the real and imaginary results for a given nodal diameter, assuming no phase shift between them; however, the modal response can occur at *any* phase shift.

CYCPHASE response results are valid only for the first cyclic sector. To obtain the response at *any* part of the expanded model, ANSYS recommends using cyclic symmetry results expansion at the phase angle obtained via **CYCPHASE**.

The phase angles returned by **CYCPHASE** contain the minimum and maximum values for USUM, SEQV and other scalar principal stress and strain quantities; however, they do not always return the true minimum and maximum values for directional quantities like UX or SX unless the values fall in the first sector.

CYCPHASE does not consider midside node values when evaluating maximum and minimum values, which may affect **DISPLAY** quantities but no others. (Typically, ANSYS ignores midside node stresses and strains during postprocessing.)

Issuing **CYCPHASE**,PUT clears the result values for midside nodes on high order elements; therefore, this option sets element faceting (**/EFACET**) to 1. The command reports that midside nodal values are set to zero and indicates that element faceting is set to 1.

If the sweep values are available after issuing a **CYCPHASE**,PUT command, the **PRNSOL** or **PLNSOL** command will print or plot (respectively) the sweep values of structure displacement Ux, Uy, Uz, component stress/strain X, Y, Z, XY, YZ, ZX, principal stress/strain 1, 2, 3 and equivalent stress/strain EQV. The vector sum of displacement (USUM) and stress/strain intensity (SINT) are not valid phase-sweep results.

The **RSYS** command has no effect on the **CYCPHASE** command. Component values reported by **CYCPHASE** are in the global Cartesian coordinate system.

The **CYCPHASE** command is valid in **/POST1** and for cyclically symmetric models only.

To learn more about analyzing a cyclically symmetric structure, see Cyclic Symmetry Analysis in the *ANSYS Advanced Analysis Techniques Guide*.

Menu Paths

Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Clear
Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Get
Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>List
Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Phase Sweep
Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Put Phase Results
Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Status

CYL4, *XCENTER*, *YCENTER*, *RAD1*, *THETA1*, *RAD2*, *THETA2*, *DEPTH*

Creates a circular area or cylindrical volume anywhere on the working plane.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

XCENTER, *YCENTER*
Working plane X and Y coordinates of the center of the circle or cylinder.

RAD1, *RAD2*
Inner and outer radii (either order) of the circle or cylinder. A value of zero or blank for either *RAD1* or *RAD2*, or the same value for both *RAD1* and *RAD2*, defines a solid circle or cylinder.

THETA1, *THETA2*
Starting and ending angles (either order) of the circle or faces of the cylinder. Used for creating a partial annulus or partial cylinder. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0° and the ending angle defaults to 360°. See the *ANSYS Modeling and Meshing Guide* for an illustration.

DEPTH
The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the cylinder. If *DEPTH* = 0 (default), a circular area is created on the working plane.

Notes

Defines a circular area anywhere on the working plane or a cylindrical volume with one face anywhere on the working plane. For a solid cylinder of 360°, the top and bottom faces will be circular (each area defined with four lines) and they will be connected with two surface areas (each spanning 180°). See the **CYL5**, **PCIRC**, and **CYLIND** commands for alternate ways to create circles and cylinders.

When working with a model imported from an IGES file (DEFAULT import option), you must provide a value for *DEPTH* or the command will be ignored.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Annulus
Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Partial Annulus
Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Solid Circle
Main Menu>Preprocessor>Modeling>Create>Primitives>Solid Cylindr
Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>Hollow Cylinder
Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>Partial Cylinder
Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>Solid Cylinder
Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Cylinder>Solid Cylinder

CYL5, *XEDGE1*, *YEDGE1*, *XEDGE2*, *YEDGE2*, *DEPTH*

Creates a circular area or cylindrical volume by end points.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

XEDGE1, *YEDGE1*
Working plane X and Y coordinates of one end of the circle or cylinder face.

XEDGE2, *YEDGE2*
Working plane X and Y coordinates of the other end of the circle or cylinder face.

DEPTH
The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the cylinder. If *DEPTH* = 0 (default), a circular area is created on the working plane.

Notes

Defines a circular area anywhere on the working plane or a cylindrical volume with one face anywhere on the working plane by specifying diameter end points. For a solid cylinder of 360°, the top and bottom faces will be circular (each area defined with four lines) and they will be connected with two surface areas (each spanning 180°). See the **CYL4**, **PCIRC**, and **CYLIND** commands for alternate ways to create circles and cylinders.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Circle>By End Points
Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>By End Pts & Z
Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Cylinder>By End Pts & Z

CYLIND, *RAD1*, *RAD2*, *Z1*, *Z2*, *THETA1*, *THETA2*

Creates a cylindrical volume centered about the working plane origin.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

RAD1, *RAD2*
Inner and outer radii (either order) of the cylinder. A value of zero or blank for either *RAD1* or *RAD2*, or the same value for both *RAD1* and *RAD2*, defines a solid cylinder.

Z1, *Z2*
Working plane Z coordinates of the cylinder. If either *Z1* or *Z2* is zero, one of the faces of the cylinder will be coplanar with the working plane.

THETA1, *THETA2*
Starting and ending angles (either order) of the cylinder. Used for creating a cylindrical sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0.0° and the ending angle defaults to 360.0°. See the *ANSYS Modeling and Meshing Guide* for an illustration.

Notes

Defines a cylindrical volume centered about the working plane origin. The top and bottom faces are parallel to the working plane but neither face need be coplanar with (i.e., "on") the working plane. The cylinder must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) For a solid cylinder of 360°, the top and bottom faces will be circular (each area defined with four lines), and they will be connected with two areas (each spanning 180°.) See the **CYL4** and **CYL5** commands for alternate ways to create cylinders.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>By Dimensions
Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Cylinder>By Dimensions

D Commands

D, *NODE*, *Lab*, *VALUE*, *VALUE2*, *NEND*, *NINC*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*
Defines DOF constraints at nodes.

SOLUTION: FE Constraints

MP ME ST DY <> PR EM <> FL PP ED

NODE
Node at which constraint is to be specified. If ALL, *NEND* and *NINC* are ignored and constraints are applied to all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab
Valid degree of freedom label. If ALL, use all appropriate labels. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). FLOTRAN fluid labels: PRES (pressure); VX, VY, or VZ (velocities); ENKE or ENDS (turbulent kinetic energy or turbulent kinetic energy dissipation rate); SP01 through SP06 (multiple species mass fractions) or their user-defined names [**MSSPEC**]; UX, UY, or UZ (Arbitrary Lagrangian-Eulerian formulation mesh displacements). Electric labels: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials).

VALUE
Degree of freedom value or table name reference for tabular boundary conditions. To specify a table, enclose the table name in percent signs (%) (e.g., **D**,NODE,TEMP,%*tabname*%). Use the ***DIM** command to define a table.

If *Lab* = ENKE and *VALUE* = -1, a FLOTRAN flag is set to indicate a moving wall.

If *Lab* = ENDS and *VALUE* = -1, FLOTRAN generalized symmetry conditions are applied. Velocity components are set tangential to the symmetry surface if the ALE formulation is not activated. They are set equal to the mesh velocity if the ALE formulation is activated.

VALUE2
Second degree of freedom value (if any). If the analysis type and the degree of freedom allow a complex input, *VALUE* (above) is the real component and *VALUE2* is the imaginary component.

NEND, *NINC*
Specifies the same values of constraint at the range of nodes from *NODE* to *NEND* (defaults to *NODE*), in steps of *NINC* (defaults to 1).

Lab2, *Lab3*, *Lab4*, *Lab5*, *Lab6*
Additional degree of freedom labels. The same values are applied to the nodes for these labels.

Notes

The available degrees of freedom per node are listed under "Degrees of Freedom" in the input table for each element type in the *ANSYS Elements Reference*. Degrees of freedom are defined in the nodal coordinate system. The positive directions of structural translations and rotations are along and about the positive nodal axes directions. Structural rotations should be input in radians. The node and the degree of freedom label must be selected [**NSEL**, **DOFSEL**].

For elements HF119 and HF120, used in high-frequency electromagnetic analysis, the AX DOF is not an x-component of a vector potential, but rather a tangential component of E (the electric field) on the element edges and faces. To specify an Electric Wall condition, set AX to zero. For more information, see the *ANSYS High-Frequency Electromagnetic Analysis Guide*.

For element SOLID117 used in static and low frequency electromagnetic analysis, the AZ DOF is not a z-component of a vector potential, but rather the flux contribution on the element edge. To specify a flux-parallel condition, set AZ = 0. For more information, see 3-D Magnetostatics and Fundamentals of Edge-based Analysis in the *ANSYS Low-Frequency Electromagnetic Analysis Guide*.

In an explicit dynamic analysis, the **D** command can only be used to fix nodes in the model. The degree of freedom value must be zero; no other values are valid. Use the **EDLOAD** command to apply a non zero displacement in an explicit dynamic analysis.

Tabular boundary conditions (*VALUE* = %`tabname`%) are available only for the following degree of freedom labels: Electric (VOLT), FLOTRAN (UX, UY, UZ, PRES, VX, VY, VZ, ENKE, ENDS, TEMP, SP01, SP02, SP03, SP04, SP05, and SP06); structural (UX, UY, UZ, ROTX, ROTY, ROTZ), and temperature (TEMP, TBOT, TE2, TE3, . . ., TTOP). All labels are valid only in static (**ANTYPE**,STATIC) and full transient (**ANTYPE**,TRANS) analyses.

%_FIX% is an ANSYS reserved table name. When *VALUE* is set to %_FIX%, ANSYS will prescribe the DOF to the "current" displacement value. Alternatively, functions UX(), UY(), etc. may be used (see ***GET** for a complete list of available functions). However, note that these functions are not available when multiframe restart features are invoked. In most cases, %_FIX% usage is efficient and recommended for all structural degrees of freedom.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PerfEC>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Node Components
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>BasePSD>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>BasePSD>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Nodes
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>On Nodes
Main Menu>Solution>Constraints>Apply>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PerfEC>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Nodes
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Node Components
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Nodes
Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>BasePSD>On Nodes
Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Nodes
Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>BasePSD>On Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Nodes

---

DA, *AREA*, *Lab*, *Value1*, *Value2*

Defines DOF constraints on areas.

SOLUTION: Solid Constraints

MP ME ST <> <> PR EM <> <> PP ED

AREA
Area on which constraints are to be specified. If ALL, apply to all selected areas [**ASEL**]. If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *AREA*.

Lab
Symmetry label (see 2 below):

SYMM
Generate symmetry constraints for non-FLOTRAN models. Requires no *Value1* or *Value2*.

ASYM
Generate antisymmetry constraints for non-FLOTRAN models. Requires no *Value1* or *Value2*.

ANSYS DOF labels (see 1, 2, and 3 below):

UX
Displacement in X direction.

UY
Displacement in Y direction.

UZ
Displacement in Z direction.

ROTX
Rotation about X axis.

ROTY
Rotation about Y axis.

ROTZ
Rotation about Z axis.

TEMP, TBOT, TE2, TE3, . . ., TTOP
Temperature.

MAG
Magnetic scalar potential.

VOLT
Electric scalar potential.

AX
Magnetic vector potential in X direction (see 4).

AY
Magnetic vector potential in Y direction.

AZ
Magnetic vector potential in Z direction (see 1).

ALL
Applies all appropriate DOF labels.

FLOTRAN Standard DOF Labels (see 7): VX, VY, VZ, PRES, TEMP, ENKE, ENDS

FLOTRAN Species Labels (See 8): SP01, SP02, SP03, SP04, SP05, SP06

FLOTRAN Arbitrary Lagrangian-Eulerian formulation Mesh Displacement Labels (See 9): UX, UY, UZ

Value1
Value of DOF or table name reference on the area. Valid for all DOF labels. To specify a table, enclose the table name in % signs (e.g., **DA**,*AREA*,TEMP,%*tabname*%). Use the ***DIM** command to define a table.

If *Lab* = ENKE and *Value1* = -1, a FLOTRAN flag is set to indicate a moving wall.

If *Lab* = ENDS and *Value1* = -1, FLOTRAN generalized symmetry conditions are applied. Velocity components are set tangential to the symmetry surface if the ALE formulation is not activated. They are set equal to the mesh velocity if the ALE formulation is activated.

Value2
For FLOTRAN DOFs:

0
Values are applied only to nodes within the area.

1
Values are applied to the edges of the area as well as to the internal nodes. (See 7.)

For MAG and VOLT DOFs:

Value of the imaginary component of the degree of freedom.

Notes

1. For element SOLID117, if *Lab* = AZ and *Value1* = 0, this sets the flux-parallel condition for the edge formulation. (A flux-normal condition is the natural boundary condition.) Do *not* use the **DA** command to set the edge-flux DOF, AZ to a nonzero value.

2. If *Lab* = MAG and *Value1* = 0, this sets the flux-normal condition for the magnetic scalar potential formulations (MSP) (A flux-parallel condition is the natural boundary condition for MSP.)
3. If *Lab* = VOLT and *Value1* = 0, the J-normal condition is set (current density (J) flow normal to the area). (A J-parallel condition is the natural boundary condition.)
4. For elements HF119 and HF120, used in high-frequency electromagnetic analysis, the AX DOF is not an x-component of a vector potential, but rather a tangential component of E (the electric field) on the element edges and faces. To specify an Electric Wall condition, set AX to zero. For more information, see the *ANSYS High-Frequency Electromagnetic Analysis Guide*.
5. You can transfer constraints from areas to nodes with the **DTRAN** or **SBCTRAN** commands. See the **DK** command for information about generating other constraints on areas for non-FLOTRAN models.
6. Symmetry and antisymmetry constraints are generated as described for the **DSYM** command.
7. For the velocity DOF (VX, VY, VZ), a zero value will override a nonzero value at the intersection of two areas.
8. You can use the **MSSPEC** command to change FLOTRAN species labels to user-defined labels. You must define these labels with the **MSSPEC** command before using them on the **DA** command.
9. Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following degree of freedom labels: Electric (VOLT), FLOTRAN (UX, UY, UZ, PRES, VX, VY, VZ, ENKE, ENDS, TEMP, SP01, SP02, SP03, SP04, SP05, and SP06); Structural (UX, UY, UZ, ROTX, ROTY, ROTZ), and temperature (TEMP, TBOT, TE2, TE3, . . ., TTOP).
10. Constraints specified by the **DA** command can conflict with other specified constraints. See Resolution of Conflicting Constraint Specifications in the *ANSYS Basic Analysis Guide* for details.
11. The **DA** command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PerfEC>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Areas
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>On Areas
Main Menu>Solution>Constraints>Apply>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PerfEC>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Areas
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Areas
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Areas
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Areas
Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Areas

DADELE, *AREA*, *Lab*

Deletes DOF constraints on an area.

SOLUTION: Solid Constraints

MP ME ST <> <> PR EM <> <> PP ED

AREA

Area for which constraints are to be deleted. If ALL, delete for all selected areas [**ASEL**]. If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). You can substitute a component name for *AREA*.

Lab

Valid constraint labels are:

ALL
: All constraints.

SYMM
: Symmetry constraints.

ASYM
: Antisymmetry constraints.

UX
: Displacement in X direction.

UY
: Displacement in Y direction.

UZ
: Displacement in Z direction.

ROTX
: Rotation about X axis.

ROTY
Rotation about Y axis.

ROTZ
Rotation about Z axis.

VX
Velocity component in X direction.

VY
Velocity component in Y direction.

VZ
Velocity component in Z direction.

PRES
Pressure.

TEMP, TBOT, TE2, TE3, . . ., TTOP
Temperature.

ENKE
Turbulent Kinetic Energy.

ENDS
Energy Dissipation Rate.

MAG
Magnetic scalar potential.

VOLT
Electric scalar potential.

SP01SP06
Multiple Species Mass Fraction.

AX
Magnetic vector potential in X direction (see notes).

AY
Magnetic vector potential in Y direction.

AZ
Magnetic vector potential in Z direction (see notes).

Notes

Deletes the degree of freedom constraints at an area (and all corresponding finite element constraints) previously specified with the **DA** command. See the **DDELE** command for delete details.

If the multiple species labels have been changed to user-defined labels via the **MSSPEC** command, use the user-defined labels.

For element SOLID117, AZ is the electromagnetic edge-flux DOF. See the **DA** command for details.

For elements HF119 and HF120, used in high-frequency electromagnetic analysis, the AX DOF is not an x-component of a vector potential, but rather a tangential component of E (the electric field) on the element edges and faces. To specify an Electric Wall condition, set AX to zero. For more information, see the *ANSYS High-Frequency Electromagnetic Analysis Guide*.

Warning: *On previously meshed areas*, all constraints on affected nodes will be deleted, whether or not they were specified by the DA command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PerfEC>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>TimeInt>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Displacement>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Species>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Turbulence>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Velocity>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Areas
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>On Areas
Main Menu>Solution>Constraints>Delete>On Areas
Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All Areas
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PerfEC>On Areas
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>TimeInt>On Areas
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Areas
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Areas
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Displacement>On Areas
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Areas
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Species>On Areas
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Turbulence>On Areas
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Velocity>On Areas
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Areas
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Areas
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Areas
Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Areas
Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Areas

---

DALIST, *AREA*

Lists the DOF constraints on an area.

SOLUTION: Solid Constraints

MP ME ST <> <> PR EM <> <> PP ED

AREA

List constraints for this area. If ALL (default), list for all selected areas [**ASEL**]. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *AREA*.

Notes

Lists the degree of freedom constraints on an area previously specified with the **DA** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>DOF Constraints>On All Areas
Utility Menu>List>Loads>DOF Constraints>On Picked Areas

DAMORPH, *AREA*, *XLINE*, *RMSHKY*

Move nodes in selected areas to conform to structural displacements.

PREP7: Morphing

MP <> <> <> <> <> <> <> <> PP ED

AREA
Non-structural area to which mesh movement (morph) applies. If ALL, apply morphing to all selected areas [ASEL]. If *AREA* = P, graphical picking is enabled. A component may be substituted for *AREA*.

XLINE
Lines to be excluded from morphing. If ALL, exclude all selected lines [LSEL] from morphing. If *XLINE* = P, graphical picking is enabled. A component may be substituted for *XLINE*. If *XLINE* is blank (default), allow morphing of nodes attached to lines of the selected areas (*AREA*) which are not shared by unselected areas. See Notes for clarification.

RMSHKY
Remesh flag option:

0
Remesh the selected non-structural areas only if mesh morphing fails.

1
Remesh the selected non-structural areas and bypass mesh morphing.

2
Perform mesh morphing only and do not remesh.

Notes

The selected areas should include only non-structural regions adjacent to structural regions. **DAMORPH** will morph the non-structural areas to coincide with the deflections of the structural regions.

Nodes in the structural regions move in accordance with computed displacements. Displacements from a structural analysis must be in the database prior to issuing **DAMORPH**.

By default, nodes attached to lines can move along the lines, or off the lines (if a line is interior to the selected areas). You can use *XLINE* to restrain nodes on certain lines.

By default (*RMSHKEY*=0), **DAMORPH** will remesh the selected non-structural areas entirely if a satisfactory morphed mesh cannot be provided.

If boundary conditions and loads are applied directly to nodes and elements, the **DAMORPH** command requires that these be removed before remeshing can take place.

Exercise care with initial conditions defined by the **IC** command. Before a structural analysis is performed for a sequentially coupled analysis, the **DAMORPH** command requires that initial conditions be removed from all null element type nodes in the non-structural regions. Use **ICDELE** to delete the initial conditions.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Phys Morphing>Areas

---

DATA, *IR*, *LSTRT*, *LSTOP*, *LINC*, *Name*, *KCPLX*

Reads data records from a file into a variable.

POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

LSTRT
Start at location *LSTRT* (defaults to 1).

LSTOP
Stop at location *LSTOP* (defaults to *LSTRT*). Maximum location available is determined from data previously stored.

LINC
Fill every *LINC* location between *LSTRT* and *LSTOP* (defaults to 1).

Name
Eight character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

KCPLX
Complex number key:

0
Data stored as the real part of the complex number.

1
Data stored as the imaginary part of the complex number.

Notes

This command must be followed by a format statement (on the next line) and the subsequent data records, and all must be on the same file (that may then be read with the **/INPUT** command). The format specifies the number of fields to be read per record, the field width, and the placement of the decimal point (if one is not included in the data value). The read operation follows the available FORTRAN FORMAT conventions of the system. See the system FORTRAN manual for details. Any standard FORTRAN real format (such as (4F6.0), (F2.0,2X,F12.0), etc.) may be used. Integer (I), character (A), and list-directed (*) descriptors may *not* be used. *The parentheses must be included in the format*. Up to 80 columns per record may be read. Locations may be filled within a range. Previous data in the range will be overwritten.

Menu Paths

This command cannot be accessed from a menu.

DATADEF

Specifies "Directly defined data status" as the subsequent status topic.

POST1: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Modify Results

DCGOMG, *DCGOX*, *DCGOY*, *DCGOZ*

Specifies the rotational acceleration of the global origin.

SOLUTION: Inertia

MP ME ST <> <> PR <> <> <> PP ED

DCGOX, *DCGOY*, *DCGOZ*

Rotational acceleration of the global origin about the acceleration system X, Y, and Z axes.

Notes

Specifies the rotational acceleration of the global origin about each of the acceleration coordinate system axes [**CGLOC**]. Rotational accelerations may be defined in analysis types **ANTYPE**,STATIC, HARMIC (full or mode superposition), TRANS (full or mode superposition), and SUBSTR. See the *ANSYS, Inc. Theory Reference* for details. Units are radians/time^2. Related commands are **ACEL**, **CGLOC**, **CGOMGA**, **DOMEGA**, and **OMEGA**.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Coriolis Effects
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Coriolis Effects
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Coriolis Effects
Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Coriolis Effects

---

DCUM, *Oper*, *RFACT*, *IFACT*, *TBASE*

Specifies that DOF constraint values are to be accumulated.

SOLUTION: FE Constraints

MP ME ST <> <> PR EM <> FL PP ED

Oper
Accumulation key:

REPL
Subsequent values replace the previous values (default).

ADD
Subsequent values are added to the previous values.

IGNO
Subsequent values are ignored.

RFACT
Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

IFACT
Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

TBASE
Base temperature for temperature difference. Used only with temperature degree of freedom. Scale factor is applied to the temperature difference (*T*-*TBASE*) and then added to *TBASE*. *T* is the current temperature.

Command Default

Replace previous values.

Notes

Allows repeated degree of freedom constraint values (displacement, temperature, etc.) to be replaced, added, or ignored. Operations apply to the selected nodes [**NSEL**] and the selected degree of freedom labels [**DOFSEL**]. The operations occur when the next degree of freedom constraints are defined. For example, issuing the command **D**,1,UX,.025 after a previous **D**,1,UX,.020 causes the new value of the displacement on node 1 in the x-direction to be 0.045 with the add operation, 0.025 with the replace operation, or 0.020 with the ignore operation. Scale factors are also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a displacement of 0.070. Scale factors are applied even if no previous values exist. Issue **DCUM**,STAT to show the current label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

Note — FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

DCUM does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Constraints
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Constraints

---

DCVSWP, *Option*, *Elem*, *Cnum*, *Vmax*, *Vinc1*, *Vinc2*, *Gap*

Performs a DC voltage sweep on a ROM element.

REDUCED ORDER MODELING: Use Pass
MP <> <> <> <> <> <> <> <> PP ED

Option
Sweep option:

GV
Perform voltage sweep up to given voltage *Vmax*.

PI
Perform a voltage sweep simulation up to the pull-in voltage.

Elem
Element number of the ROM element for the ROM use pass analysis.

Cnum
Number of sweep conductor.

Vmax
Maximum voltage. For the PI option, this voltage should be below the pull-in voltage value.

Vinc1
Voltage increment for Vmax (default = *Vmax*/20).

Vinc2
Voltage increment for pull-in voltage (default = 1).

Gap
Gap elements option:

0
Create gap elements (COMBIN40) (default).

1
Do not create gap elements

Notes

Vinc1 is used to ramp the sweep conductor voltage from 0 to *Vmax*. *Vinc2* is used to increase the sweep conductor voltage from *Vmax* to the pull-in value if the PI sweep option is used.

Because ramping the voltage may lead to the unstable region of an electromechanical system, **DCVSWP** might not converge when the sweep conductor voltage approaches the pull-in value. To avoid non-converged solutions, you should use the gap option to create a set of spring-gap elements (COMBIN40). By default, **DCVSWP** creates two spring-gap elements with opposite orientations for each active modal displacement DOF of the ROM element. The gap size is set to the maximum absolute values of the deflection range for the corresponding mode, as cal-

culated by **RMMSELECT** or modified using the **RMMRANGE** command. The spring constants are set to 1.E5 for all the COMBIN40 elements. Along with the spring-gap elements, **DCVSWP** creates a set of constraint equations relating the ROM element modal displacements DOF (EMF) and the displacement DOF (UX) of the gap elements. Constraining the modal displacements using the spring-gap elements allows **DCVSWP** to converge in the pull-in range. The **DCVSWP** macro has a limit of 900 equilibrium iterations. If this limit is not sufficient to reach convergence, try the advanced techniques given in Overcoming Convergence Problems in the *ANSYS Structural Analysis Guide*. For more information on gap elements, see Using Gap Elements with ROM144 in the *ANSYS Coupled-Field Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>ROM Tools>Voltage Sweep
Main Menu>Solution>ROM Tools>Voltage Sweep

DDELE, *NODE*, *Lab*, *NEND*, *NINC*

Deletes degree of freedom constraints.

SOLUTION: FE Constraints

MP ME ST DY <> PR EM <> FL PP ED

NODE
Node for which constraint is to be deleted. If ALL, *NEND* and *NINC* are ignored and constraints for all selected nodes [**NSEL**] are deleted. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab
Valid degree of freedom label. If ALL, use all selected labels [**DOFSEL**]. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). FLOTRAN fluid labels: PRES (pressure); VX, VY, or VZ (velocities); ENKE or ENDS (turbulent kinetic energy or turbulent energy dissipation); SP01 through SP06 (multiple species mass fractions) or their user-defined names. Electric label: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials). High-frequency electromagnetic label: AX (Electric Wall or Magnetic Wall boundary condition).

NEND, *NINC*
Delete constraints from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1).

Notes

Deleting a constraint is not the same as setting it to zero (which "fixes" the degree of freedom to a zero value). Deleting a constraint has the same effect as deactivating, releasing, or setting the constraint "free." The node and the degree of freedom label must be selected [**NSEL**, **DOFSEL**].

For elements HF119 and HF120, used in high-frequency electromagnetic analysis, the AX DOF is not an x-component of a vector potential, but rather a tangential component of E (the electric field) on the element edges and faces. To specify an Electric Wall condition, set AX to zero. For more information, see the *ANSYS High-Frequency Electromagnetic Analysis Guide*.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PerfEC>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>TimeInt>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Displacement>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Species>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Turbulence>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Velocity>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Node Components
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Nodes
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>On Nodes
Main Menu>Solution>Constraints>Delete>On Nodes
Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PerfEC>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>TimeInt>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Nodes
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Nodes
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Displacement>On Nodes
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Nodes
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Species>On Nodes
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Turbulence>On Nodes
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Velocity>On Nodes
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Nodes
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Nodes
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Node Components
Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Nodes
Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Nodes

DEACT

Specifies "Element birth and death" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Elem Birth/Death

DECOMP, --, Ndomains

Decomposes the model into domains used by distributed solvers (DPCG, DJCG, DDS)

PREP7: Special Purpose

MP ME ST DY <> PR EM <> FL PP ED

--

Unused field

Ndomains

Sets the number of preconditioner domains for model decomposition.

AUTO

Automatically set the number of domains (default).

N

Set the number of domains to be used by the preconditioner of the solver to *N*>1.

Command Default

DECOMP,,*AUTO*

Notes

This command breaks (decomposes) the model into domains for previewing purposes through the **/PNUM** and **EPLOT** commands. See Improving ANSYS Performance and Parallel Performance for ANSYS in the *ANSYS Advanced Analysis Techniques Guide*

If you specify **DECOMP** (no arguments), the command defaults to **DECOMP**,,AUTO. The domains decomposed here are the domains that are used by a solver's preconditioners only. The larger domains built based on these "preconditioner" domains are called CPU domains. The number of CPU is set to be equal to the number of processors specified by **DSPROC**.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Domain Decomp
Main Menu>Solution>Define Loads>Domain Decomp

DEFINE

Specifies "Data definition settings" as the subsequent status topic.

POST1: Status
POST26: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Read Options
Utility Menu>List>Status>TimeHist Postproc>Variables

DELETE, SET, *Nstart*, *Nend*

Specifies sets in the results file to be deleted before postprocessing.

AUX3: Results Files

MP ME ST DY <> PR EM <> FL PP ED

SET
Specifies that sets in the results file are to be deleted.

Nstart
The first set in a results file to be deleted.

Nend
The final set in a results file to be deleted. This field is used only if deleting more than one sequential sets.

Notes

This command is valid only at the Begin Level.

Menu Paths

This command cannot be accessed from a menu.

/DELETE, *Fname*, *Ext*, --

Deletes a file.

SESSION: Files

MP ME ST DY <> PR EM <> FL PP ED

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to the current *Jobname*.

Ext

Filename extension (8 character maximum).

--

Unused field

Menu Paths

Utility Menu>File>File Operations>Delete
Utility Menu>PlotCtrls>Redirect Plots>To GRPH File
Utility Menu>PlotCtrls>Redirect Plots>To HPGL File
Utility Menu>PlotCtrls>Redirect Plots>To HPGL2 File
Utility Menu>PlotCtrls>Redirect Plots>To PSCR File

DELTIM, *DTIME*, *DTMIN*, *DTMAX*, *Carry*

Specifies the time step sizes to be used for this load step.

SOLUTION: Load Step Options

MP ME ST <> <> PR EM <> <> PP ED

DTIME

Time step size for this step. If automatic time stepping is being used [**AUTOTS**], *DTIME* is the starting time substep. If **SOLCONTROL**,ON and contact elements TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, or CONTA174 are used, defaults to 1 or 1/20 the total time span of the load step, depending on the physics of the problem. If **SOLCONTROL**,ON and none of these contact elements are used, defaults to 1 time span of the load step. If **SOLCONTROL**,OFF, defaults to the previously specified value.

DTMIN

Minimum time step (if automatic time stepping is used). If **SOLCONTROL**,ON, default determined by ANSYS depending on the physics of the problem. If **SOLCONTROL**,OFF, defaults to the previously specified value (or *DTIME*, if there is no previously specified value).

DTMAX

Maximum time step (if automatic time stepping is used). If **SOLCONTROL**,ON, default determined by ANSYS depending on the physics of the problem. If **SOLCONTROL**,OFF, defaults to the previously specified value (or the time span of the load step, if there is no previously specified value).

Carry

Time step carry over key:

OFF

Use *DTIME* as time step at start of each load step.

ON
Use final time step from previous load step as the starting time step (if automatic time stepping is used).

If **SOLCONTROL**,ON, default determined by ANSYS depending on the physics of the problem. If **SOLCONTROL**,OFF, defaults to OFF.

Notes

See **NSUBST** for an alternative input. Use values for *DTIME* and *TIME* [**TIME**] that are consistent. For example, using 0.9 for *DTIME* and 1.0 for *TIME* will result in one time step because 1.0 (*TIME*) is divisible by .9 (*DTIME*) at most once. If your intent is to load in 10 increments over a time span of 1.0, then use 0.1 for *DTIME* and 1.0 for *TIME*. It is recommended that all fields of this command be specified for solution efficiency and robustness.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step
Main Menu>Solution>Analysis Type>Sol'n Controls>Basic
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step

---

DEMORPH, *ELEM*, *DIMN*, *RMSHKY*

Move nodes in selected elements to conform to structural displacements.

PREP7: Morphing

MP <> <> <> <> <> <> <> <> PP ED

ELEM
Non-structural elements to which mesh movement (morph) applies. If ALL, apply morphing to all selected elements [ESEL]. If ELEM = P, graphical picking is enabled. A component may be substituted for ELEM.

DIMN
Problem dimensionality. Use "2" for a 2-D problem and "3" for a 3-D problem (no default).

RMSHKY
Remesh flag option:

0
Remesh the selected non-structural regions only if mesh morphing fails.

1
Remesh the selected non-structural regions and bypass mesh morphing.

2
Perform mesh morphing only and do not remesh.

Notes

The selected elements should include only non-structural regions adjacent to structural regions. The exterior nodes of the selected elements will usually be on the boundary of the region which will have node positions displaced. For *DIMN* = 2, elements must lie on a flat plane. The **DEMORPH** command requires a single domain grouping of elements be provided (multiple domains of elements are not permitted). Exterior nodes will be as-

sumed fixed (no nodes will be morphed) unless they coincide with structural nodes having nonzero displacements.

Nodes in the structural regions move in accordance with computed displacements. Displacements from a structural analysis must be in the database prior to issuing **DEMORPH**.

By default (*RMSHKY*=0), **DEMORPH** will remesh the selected non-structural regions entirely if a satisfactory morphed mesh cannot be provided.

If boundary conditions and loads are applied directly to nodes and elements, the **DEMORPH** command requires that these be removed before remeshing can take place.

Exercise care with initial conditions defined by the **IC** command. Before a structural analysis is performed for a sequentially coupled analysis, the **DEMORPH** command requires that initial conditions be removed from all null element type nodes in the non-structural regions. Use **ICDELE** to delete the initial conditions.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Phys Morphing>Elements

---

DERIV, *IR*, *IY*, *IX*, --, *Name*, --, --, *FACTA*

Differentiates a variable.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IY, *IX*
Reference numbers of variables to be operated on. *IY* is differentiated with respect to *IX*.

--
Unused field.

Name
Thirty-two character name for identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --
Unused fields.

FACTA
Scaling factor (positive or negative) applied as shown below (defaults to 1.0).

Notes

Differentiates variables according to the operation:

$$IR = FACTA \times d(IY)/d(IX)$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Derivative

DESIZE, *MINL*, *MINH*, *MXEL*, *ANGL*, *ANGH*, *EDGMN*, *EDGMX*, *ADJF*, *ADJM*

Controls default element sizes.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

MINL
Minimum number of elements that will be attached to a line when using lower-order elements (defaults to 3 elements per line). If *MINL* = DEFA, all arguments will be set back to default values. If *MINL* = STAT, list status of command (Including on/off status). If *MINL* = OFF, deactivate default element sizing. If *MINL* = ON, reactivate default element sizing.

MINH
Minimum number of elements that will be attached to a line when using higher-order elements (defaults to 2 elements per line).

MXEL
Maximum number of elements that will be attached to a single line (lower or higher-order elements) (defaults to 15 elements per line for h-elements and 6 divisions per line for p-elements).

ANGL
Maximum spanned angle per lower-order element for curved lines (defaults to 15 degrees per element).

ANGH
Maximum spanned angle per higher-order element for curved lines (defaults to 28 degrees per element).

EDGMN
Minimum element edge length (defaults to no minimum edge length).

EDGMX
Maximum element edge length (defaults to no maximum edge length).

ADJF
Target aspect ratio for adjacent line. Used only when free meshing (defaults to 1.0, which attempts to create equal sided h-elements; defaults to 4 for p-elements).

ADJM
Target aspect ratio for adjacent line. Used only when map meshing (defaults to 4.0, which attempts to create rectangular h-elements; defaults to 6 for p-elements).

Command Default

Default settings as described for each argument are used.

Notes

DESIZE settings are usually used for mapped meshing. They are also used for free meshing if SmartSizing is turned off [**SMRTSIZE**,OFF], which is the default. Even when SmartSizing is on, some **DESIZE** settings (such as maximum and minimum element edge length) can affect free mesh density. The default settings of the **DESIZE** command are used only when no other element size specifications [**KESIZE**, **LESIZE**, **ESIZE**] exist for a certain line.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Other

DESOL, *ELEM*, *NODE*, *Item*, *Comp*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

Defines or modifies solution results at a node of an element.

POST1: Set Up

MP ME ST DY <> PR EM <> <> PP ED

ELEM
Element number for which results are defined or modified. If ALL, apply to all selected elements [**ESEL**].

NODE
Node of element (actual node number, not the position) to which results are specified. If ALL, specify results for all selected nodes [**NSEL**] of element. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Item
Label identifying results. Valid item labels are shown in DESOL - Valid Item and Component Labels below. Some items also require a component label (*Comp*).

Comp
Component of the item (if required); see DESOL - Valid Item and Component Labels.

V1
Value (in the element coordinate system) assigned to the database item (and component, if any). If zero, a zero value will be assigned. If blank, value remains unchanged.

V2, *V3*, *V4*, *V5*, *V6*
Additional values (if any) assigned to the remaining components (in the order corresponding to the *Comp* list shown below) for the specified *Item* (starting from the specified *Comp* label and proceeding to the right).

Notes

Defines or modifies solution results in the database at a node of an area or volume element. For example, **DESOL**,35,50,S,X,1000,2000,1000 assigns values 1000, 2000, and 1000 to SX, SY, and SZ (respectively) of node 50 of element 35. The settings of the POST1 **FORCE**, **SHELL**, and **LAYER** commands, if applicable, further specify which database items are affected. All data is stored in the solution coordinate system but will be displayed in the results coordinate system [**RSYS**]. Use the **PRESOL** command to list the current results.

Result items are available depending on element type; check the individual element for availability. Valid item and component labels for element results are:

DESOL - Valid Item and Component Labels

Item	Comp	Description
ELEM		Element number.
S	X, Y, Z, XY, YZ, XZ	Component stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.

Item	Comp	Description
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
TG	X, Y, Z	Component thermal gradient.
TF	X, Y, Z	Component thermal flux.
PG	X, Y, Z	Component pressure gradient.
EF	X, Y, Z	Component electric field.
D	X, Y, Z	Component electric flux density.
H	X, Y, Z	Component magnetic field intensity.
B	X, Y, Z	Component magnetic flux density.
FMAG	X, Y, Z	Component magnetic force.
F	X, Y, Z	X, Y, or Z structural force.
M	X, Y, Z	X, Y, or Z structural moment.
HEAT		Heat flow.
FLOW		Fluid flow.
AMPS		Current flow.
FLUX		Magnetic flux.
VF	X, Y, Z	X, Y, or Z fluid force component.
CSG	X, Y, Z	X, Y, or Z magnetic current segment component.

Menu Paths

Main Menu>General Postproc>Define/Modify>Elem Results

DETAB, *ELEM*, *Lab*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

Modifies element table results in the database.

POST1: Set Up

MP ME ST DY <> PR EM <> FL PP ED

ELEM

Element for which results are to be modified. If ALL, modify all selected elements [**ESEL**] results. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *ELEM*.

Lab

Label identifying results. Valid labels are as defined with the **ETABLE** command. Issue **ETABLE**,STAT to display labels and values.

V1
Value assigned to this element table result in the database. If zero, a zero value will be assigned. If blank, value remains unchanged.

V2, *V3*, *V4*, *V5*, *V6*
Additional values (if any) assigned to consecutive element table columns.

Notes

Modifies element table [**ETABLE**] results in the database. For example, **DETAB**,35,ABC,1000,2000,1000 assigns 1000, 2000, and 1000 to the first three table columns starting with label ABC for element 35. Use the **PRETAB** command to list the current results. After deleting a column of data using **ETABLE**,*Lab*,ERASE, the remaining columns of data are not shifted to compress the empty slot. Therefore, the user must allocate null (blank) values for *V1*, *V2*...*V6* for any ETABLE entries which have been deleted by issuing **ETABLE**,*Lab*,ERASE. All data are stored in the solution coordinate system but will be displayed in the results coordinate system [**RSYS**].

Menu Paths

Main Menu>General Postproc>Define/Modify>ElemTabl Data

/DEVDISP, *Label*, *KEY*

Controls graphics device options.

DISPLAY: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Label
Device function label:

BBOX
Disables display information sorting for PowerGraphics displays. When activated (*KEY* = 1 or ON), model rotations and replots are performed without recalculating edge and surface data. This will speed up the rotation (especially for 2-D displays) of large models, although the display information will not be resolved as quickly (you will see a bounding box instead of the model during dynamic rotations). The default is OFF (*KEY* = 0).

DITHER
Dithering. When turned on (default), dithering smooths transitions in color intensity. Applies only to Z-buffered displays.

FONT
Font selection for the ANSYS graphics window. When *Label* = FONT, the command format is: **/DEVDISP**,FONT,*KEY*,*Val1*,*Val2*,*Val3*,*VAL4*,*Val5*,*Val6*, where *KEY* determines the type of font being controlled, and values 1 through 6 control various font parameters. *Note that these values are device specific; using the same command input file* [**/INPUT**] *on different machines may yield different results.*The following *KEY* values determine the font information that will be supplied to the appropriate driver (e.g., Postscript, X11, Win32, JPEG, ...):

KEY= 1
The command controls the LEGEND (documentation column) font.

KEY= 2
The command controls the ENTITY (node and keypoint number) font.

KEY= 3
The command controls the ANNOTATION/GRAPH font.

UNIX: Values 1 through 4 are used to find a match in the X11 database of font strings. Values 1, 2, and 3 are character strings; value 4 is a nonzero integer:

Val1
Family name (e.g., Courier*New). Substitute an asterisk (*) for any blank character that appears in a family name. If *Val1* = MENU, all other values are ignored, and a font selection menu appears (GUI must be active).

Val2
Weight (e.g., medium)

Val3
Slant (e.g., r)

Val4
Pixel size (e.g., 14). Note that this value does not affect the annotation fonts (*KEY* = 3). Use the **/TSPEC** command to control the pixel size of your annotation fonts.

Val5
unused

Val6
unused

PC: The values are encoded in a PC logical font structure. Value 1 is a character string, and the remaining values are integers:

Val1
Family name (e.g., Courier*New) Substitute an asterisk (*) for any blank character that appears in a family name. If *Val1* = MENU, all other values are ignored and a font selection menu appears (GUI must be active). A value containing all blank characters causes ANSYS to use the first available resource it finds.

Val2
Weight (0 - 1000)

Val3
Orientation (in tenths of a degree)

Val4
Height (in logical units) Note that this value does not affect the annotation fonts (*KEY* =3). Use the **/TSPEC** command to control the height of your annotation fonts.

Val5
Width (in logical units)

Val6
Italics (0 = OFF, 1 = ON)

TEXT
Text size specification for the ANSYS Graphics window. When Label = TEXT, the command format is: **/DEVDISP**,TEXT,*KEY*,*PERCENT*, where *KEY* determines the type of text being controlled (1 for LEGEND, and 2 for ENTITY), and *PERCENT* specifies the new text size as a percent of the default text size. If *PERCENT* = 100, the new text size is precisely the default size. If *PERCENT* = 200, the new text size is twice the default text size.

KEY
Control key:

OFF or 0
Turns specified function off.

ON or 1
Turns specified function on.

Command Default

Dithering on.

Menu Paths

It is part of the DISPLAY program.

/DEVICE, *Label*, *KEY*
Controls graphics device options.

GRAPHICS: Set Up
MP ME ST DY <> PR EM <> FL PP ED

Label
Device function label:

BBOX
Bounding box mode. For PowerGraphics plots involving elements with **/SHOW**,x11 and **/SHOW**,win32, ANSYS generally displays dynamic rotations faster. If KEY = 1 (ON), then a bounding box (not the elements) encompassing the model is displayed and rotated, rather than the element outlines (ON is default in preprocessing). When KEY = 0 (OFF), then dynamic rotations may be slower (ANSYS redraws the element outlines) for plots involving elements with **/SHOW**,x11 and **/SHOW**,win32. OFF is default in postprocessing. This command is ignored if **/EDGE**,WN,1 is set for any WN. This is ignored in POST1 and SOLUTION plots.

For any PowerGraphics plots involving elements, regardless of **/SHOW** settings, plots will generally be displayed faster.

VECTOR
Vector mode. In vector mode, areas, volumes, elements, and postprocessing display geometries are shown as outlines (wireframes). When vector mode is off (default), these entities are shown filled with color.

DITHER
When dithering is turned on (default), color intensity transitions are smoothed. This selection a

applies only to smooth-shaded images, i.e., Z-buffered [**/TYPE**], or raster plots with Gouraud or Phong shading [**/SHADE**].

ANIM
Select the animation type used on 2-D devices on the PC platform. A *KEY* value of BMP (or 0) sets animation mode to ANSYS Animation Controller (default). A *KEY* value of AVI (or 2) sets animation mode to AVI movie player file.

FONT

Font selection for the ANSYS graphics window. When *Label* = FONT, the command format is: **/DEVICE**,FONT,*KEY*,*Val1*,*Val2*,*Val3*,*Val4*,*Val5*,*Val6* where *KEY* determines the type of font being controlled, and values 1 through 6 control various font parameters. *Note that these values are device specific; using the same command input file* [**/INPUT**] *on different machines may yield different results.*. The following *KEY* values determine the font information that will be supplied to the appropriate driver (e.g., Postscript, X11, Win32, JPEG, ...):

KEY= 1

The command controls the LEGEND (documentation column) font.

KEY= 2

The command controls the ENTITY (node and keypoint number) font.

KEY= 3

The command controls the ANNOTATION/GRAPH font.

UNIX: Values 1 through 4 are used to find a match in the X11 database of font strings. Values 1, 2, and 3 are character strings; value 4 is a nonzero integer:

Val1

Family name (e.g., Courier). If *Val1* = MENU, all other values are ignored and a font selection menu appears (GUI must be active).

Val2

Weight (e.g., medium)

Val3

Slant (e.g., r)

Val4

Pixel size (e.g., 14). Note that this value does no affect the annotation fonts (*KEY* = 3). Use the **/TSPEC** command for annotation font size.

Val5

unused

Val6

unused

PC: The values are encoded in a PC logical font structure. Value 1 is a character string, and the remaining values are integers:

Val1

Family name (e.g., Courier*New) Substitute an asterisk (*) for any blank character that appears in a family name. If *Val1* = MENU, all other values are ignored and a font selection menu appears (GUI must be active). When this value is blank ANSYS uses the first available resource it finds.

Val2

Weight (0 - 1000)

Val3

Orientation (in tenths of a degree)

Val4

Height (in logical units)

Val5
Width (in logical units)

Val6
Italics (0 = OFF, 1 = ON)

TEXT
Text size specification for the ANSYS Graphics window. Using this label with the **/DEVICE** command requires the following form: **/DEVICE**,TEXT,*KEY*,*PERCENT*. *KEY* = 1 for LEGEND fonts; *KEY* = 2 for ENTITY fonts. *PERCENT* specifies the new text size as a percent of the default text size. If *PERCENT* = 100, the new text size is precisely the default size. If *PERCENT* = 200, the new text size is twice the default text size.

KEY
Control key:

OFF or 0
Turns specified function off.

ON or 1
Turns specified function on or designates the LEGEND font.

2
Designates the ENTITY font.

3
Designates the ANNOTATION/GRAPH font.

Command Default

Vector mode off (i.e., raster mode); dithering on.

Notes

This command is valid in any processor.

The **/DEVICE**,BBOX command is ignored in POST1 and SOLUTION plots. Also, the elements are displayed and rotated if you use **/DEVICE**,BBOX,ON and **/EDGE**,WN,1,ANGLE (effectively ignoring the BBOX option).

Menu Paths

Utility Menu>PlotCtrls>Device Options
Utility Menu>PlotCtrls>Font Controls>Entity Font
Utility Menu>PlotCtrls>Font Controls>Legend Font
Utility Menu>PlotCtrls>Redirect Plots>To PSCR File

DIG, *NODE1*, *NODE2*, *NINC*

Digitizes nodes to a surface.

PREP7: Digitizing

MP ME ST DY <> PR EM <> FL PP ED

NODE1, *NODE2*, *NINC*
Digitize nodes *NODE1* through *NODE2* in steps of *NINC*. *NODE2* defaults to *NODE1* and *NINC* defaults to 1.

Command Default

No surface digitizing.

Notes

Digitizes nodes to the surface defined by the **DSURF** command. The nodes indicated must be digitized from the tablet after this command is given. The program must be in the interactive mode and the graphics terminal show option [**/SHOW**] must be active. The global Cartesian coordinates of the nodes are stored.

Menu Paths

Main Menu>Preprocessor>Create>Nodes>Digitize Nodes>Digitize Nodes

---

DIGIT

Specifies "Node digitizing" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utilty Menu>List>Status>Preprocessor>Digitize Module

DISPLAY

Specifies "Display settings" as the subsequent status topic.

POST1: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Plot Results

/DIST, *WN*, *DVAL*, *KFACT*

Specifies the viewing distance for magnifications and perspective.

GRAPHICS: Views

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

DVAL
Distance along the view line from the observer to the focus point (defaults to value producing full-window display). Distances "too close" to the object will produce excessive magnifications. If *DVAL* = AUTO, zero, or blank, the program will calculate the distance automatically. If *DVAL* = USER, the distance of last display will be used (useful when last display automatically calculated distance).

KFACT
DVAL interpretation key:

0
Interpret numerical *DVAL* values as described above.

1
Interpret *DVAL* as a multiplier on the current distance (*DVAL* of 2 gives twice the current distance; 0.5 gives half the current distance, etc.).

Command Default

Distance is automatically calculated to produce full window magnification.

Notes

The scale factor is relative to the window shape. For example, for objects centered in a square window and with parallel projection (no perspective), a distance of ℓ /2 (+10%) produces a full window magnification, where ℓ is the largest in-plane vertical or horizontal dimension. See also **/AUTO** and **/USER** commands.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate
Utility Menu>PlotCtrls>View Settings>Magnification

DJ, *ELEM*, *LABEL*, *VALUE*

Specify displacement (or rotation) boundary conditions on the components of relative motion of a joint element.

SOLUTION: FE Constraints

MP ME ST DY <> PR EM <> FL PP ED

ELEM
Element number or ALL to be specified.

LABEL
All valid labels:

UX
Displacement in local x direction.

UY
Displacement in local y direction.

UZ
Displacement in local zdirection.

ROTX
Rotation about local x axis.

ROTY
Rotation about local y axis.

ROTZ
Rotation about local y axis.

VALUE
Value of the label.

Notes

See **DJDELE** for information on deleting displacement boundary conditions.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Joint Elems
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Joint Elems

---

DJDELE, *ELEM*, *LAB*

Deletes displacement (or rotation) boundary conditions on the components of relative motion of a joint element.

SOLUTION: FE Constraints

MP ME ST DY <> PR EM <> FL PP ED

ELEM
Element number or ALL. ALL (or leaving this blank) will delete all joint element displacement or rotational DOFs specified in *LAB*.

LAB
Valid labels are:

UX
Displacement in local x direction.

UY
Displacement in local y direction.

UZ
Displacement in local z direction.

ROTX
Rotation about local x axis.

ROTY
Rotation about local y axis.

ROTZ
Rotation about local z axis.

ALL, or (blank)
Delete all valid DOFs

Notes

See **DJ** for information on specifying displacment or rotational boundary conditions.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On Joint Elems
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Joint Elems
Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On Joint Elems
Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Joint Elems

---

DJLIST, *Elem*
Lists boundary conditions applied to joint elements.

SOLUTION: FE Constraints
MP ME ST <> <> PR EM <> <> PP ED

Elem
Element number or ALL (or blank). Lists joint element boundary conditions on the specified element(s).

Notes

Valid for MPC184. See **DJ** for information on specifying displacement or rotational boundary conditions.

Menu Paths

Utility Menu>List>Loads>Joint Element DOF Constraints>On Picked Element

---

DK, *KPOI*, *Lab*, *VALUE*, *VALUE2*, *KEXPND*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*
Defines DOF constraints at keypoints.

SOLUTION: Solid Constraints

MP ME ST <> <> PR EM <> <> PP ED

KPOI
Keypoint at which constraint is to be specified. If ALL, apply to all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Lab
Valid degree of freedom label. If ALL, use all appropriate labels. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Electric labels: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials).

VALUE
Degree of freedom value or table name reference for tabular boundary conditions. To specify a table, enclose the table name in percent signs (%) (e.g., **DK**,NODE,TEMP,%*tabname*%). Use the ***DIM** command to define a table.

VALUE2
Second degree of freedom value (if any). If the analysis type and the degree of freedom allow a complex input, *VALUE* (above) is the real component and *VALUE2* is the imaginary component.

KEXPND
Expansion key:

0
Constraint applies only to the node at this keypoint.

1
Flags this keypoint for constraint expansion.

Lab2, *Lab3*, *Lab4*, *Lab5*, *Lab6*
Additional degree of freedom labels. The same values are applied to the keypoints for these labels.

Notes

A keypoint may be flagged using *KEXPND* to allow its constraints to be expanded to nodes on the attached solid model entities having similarly flagged keypoint constraints. Constraints are transferred from keypoints to nodes with the **DTRAN** or **SBCTRAN** commands. The expansion uses interpolation to apply constraints to the nodes on the lines between flagged keypoints. If all keypoints of an area or volume region are flagged *and the constraints (label and values) are equal*, the constraints are applied to the interior nodes of the region. See the **D** command for a description of nodal constraints.

Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following degree of freedom labels: Electric (VOLT), structural (UX, UY, UZ, ROTX, ROTY, ROTZ), and temperature (TEMP, TBOT, TE2, TE3, . . ., TTOP).

Constraints specified by the **DK** command can conflict with other specified constraints. See Resolution of Conflicting Constraint Specifications in the *ANSYS Basic Analysis Guide* for details.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>BasePSD>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>BasePSD>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Keypoints
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>On Keypoints
Main Menu>Solution>Constraints>Apply>On Keypoints
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>On Keypoints
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Keypoints
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Keypoints
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Keypoints
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Keypoints
Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>BasePSD>On Keypoints
Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Keypoints
Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>BasePSD>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Keypoints

---

DKDELE, *KPOI*, *Lab*

Deletes DOF constraints at a keypoint.

SOLUTION: Solid Constraints

MP ME ST <> <> PR EM <> <> PP ED

KPOI

Keypoint for which constraint is to be deleted. If ALL, delete for all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Lab

Valid degree of freedom label. If ALL, use all appropriate labels. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). FLOTRAN fluid labels: PRES (pressure); VX, VY, or VZ (velocities); ENKE or ENDS (turbulent kinetic energy or turbulent energy dissipation); SP01 through SP06 (multiple species mass fractions) or their user-defined names. Electric label: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials). High-frequency electromagnetic label: AX (Electric Wall or Magnetic Wall boundary condition).

Notes

Deletes the degree of freedom constraints (and all corresponding finite element constraints) at a keypoint. See the **DDELE** command for details.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All KPs
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>TimeInt>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Displacement>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Velocity>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Keypoints
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>On Keypoints
Main Menu>Solution>Constraints>Delete>On Keypoints
Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All KPs
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>TimeInt>On Keypoints
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Keypoints
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Keypoints
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Displacement>On Keypoints
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Keypoints
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Velocity>On Keypoints
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Keypoints
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Keypoints
Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Keypoints

---

DKLIST, *KPOI*

Lists the DOF constraints at keypoints.

SOLUTION: Solid Constraints

MP ME ST <> <> PR EM <> <> PP ED

KPOI

List constraints for this keypoint. If ALL (default), list for all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Notes

Listing applies to the selected keypoints [**KSEL**] and the selected degree of freedom labels [**DOFSEL**].

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>DOF Constraints>On All Keypoints
Utility Menu>List>Loads>DOF Constraints>On Picked KPs

DL, *LINE*, *AREA*, *Lab*, *Value1*, *Value2*

Defines DOF constraints on lines.

SOLUTION: Solid Constraints

MP ME ST <> <> PR EM <> <> PP ED

LINE

Line at which constraints are to be specified. If ALL, apply to all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*

AREA

Area containing line. The normal to the symmetry or antisymmetry surface is assumed to lie on this area. Defaults to the lowest numbered selected area containing the line number.

Lab

Symmetry label (see 2):

SYMM
: Generate symmetry constraints for non-FLOTRAN models.

ASYM
: Generate antisymmetry constraints for non-FLOTRAN models.

ANSYS DOF labels (see 3, 4, and 5):

UX
: Displacement in X direction.

UY
: Displacement in Y direction.

UZ
: Displacement in Z direction.

ROTX
: Rotation about X axis.

ROTY
: Rotation about Y axis.

ROTZ
: Rotation about Z axis.

WARP
: Warping magnitude.

TEMP, TBOT, TE2, TE3, . . ., TTOP
: Temperature

VOLT
: Electric scalar potential.

AX
: Magnetic vector potential in X direction.

AY
: Magnetic vector potential in Y direction.

AZ
Magnetic vector potential in Z direction.

ALL
Applies all appropriate DOF labels.

FLOTRAN standard DOF labels (see 3): VX, VY, VZ, PRES, TEMP, ENKE, ENDS

FLOTRAN Species Labels (see 4): SP01, SP02, SP03, SP04, SP05, SP06

FLOTRAN Arbitrary Lagrangian-Eulerian formulation Mesh Displacement Labels (see 6): UX, UY, UZ

Value1
Value of DOF (real part) or table name reference on the line. Valid for all DOF labels. To specify a table, enclose the table name in % signs (e.g., **DL**,*LINE*,*AREA*,TEMP,%*tabname*%). Use the ***DIM** command to define a table.

If *Lab* = ENKE and *Value1* = -1, a FLOTRAN flag is set to indicate a moving wall.

If *Lab* = ENDS and *Value1* = -1, FLOTRAN generalized symmetry conditions are applied. Velocity components are set tangential to the symmetry surface if the ALE formulation is not activated. They are set equal to the mesh velocity if the ALE formulation is activated.

Value2
For FLOTRAN DOFs:

0
Values are applied only to nodes within the line.

1
Values are applied to the endpoints of the line as well as to the internal nodes (see 3).

For VOLT DOFs:

Actual value of the imaginary component of the degree of freedom.

Notes

1. You can transfer constraints from lines to nodes with the **DTRAN** or **SBCTRAN** commands. See the **DK** command for information about generating other constraints at lines.
2. Symmetry and antisymmetry constraints are generated as described on the **DSYM** command.
3. For the velocity DOF (VX, VY, VZ), a zero value will override a nonzero value at the intersection of two lines.
4. You can use the **MSSPEC** command to change FLOTRAN species labels to user-defined labels. You must define these labels with the **MSSPEC** command before using them on the **DL** command.
5. Setting *Lab* = VOLT and *Value1* = 0 applies the J-normal boundary condition (current density vector (J) flows normal to the line). No input is required for the J-parallel condition because it is the natural boundary condition.
6. Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following degree of freedom labels: Electric (VOLT), FLOTRAN (UX, UY, UZ, PRES, VX, VY, VZ, ENKE, ENDS, TEMP, SP01, SP02, SP03, SP04, SP05, and SP06); Structural (UX, UY, UZ, ROTX, ROTY, ROTZ), and temperature (TEMP, TBOT, TE2, TE3, . . ., TTOP).

7. Constraints specified by the **DL** command can conflict with other specified constraints. See Resolution of Conflicting Constraint Specifications in the *ANSYS Basic Analysis Guide* for details.
8. This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PerfEC>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>...with Area
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>...with Area
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Lines
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>...with Area
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>On Lines
Main Menu>Solution>Constraints>Apply>...with Area
Main Menu>Solution>Constraints>Apply>On Lines
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PerfEC>On Lines
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Lines
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Lines
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Lines
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Lines
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>...with Area
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Lines
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Lines
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>...with Area
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Lines
Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Lines

---

DLDELE, *LINE*, *Lab*

Deletes DOF constraints on a line.

SOLUTION: Solid Constraints

MP ME ST <> <> PR EM <> <> PP ED

LINE

Line for which constraints are to be deleted. If ALL, delete for all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*

Lab
Constraint label:

ALL
All constraints.

SYMM
Symmetry constraints.

ASYM
Antisymmetry constraints.

UX
Displacement in X direction.

UY
Displacement in Y direction.

UZ
Displacement in Z direction.

ROTX
Rotation about X axis.

ROTY
Rotation about Y axis.

ROTZ
Rotation about Z axis.

WARP
Warping magnitude.

VX
Velocity component in X direction.

VY
Velocity component in Y direction.

VZ
Velocity component in Z direction.

PRES
Pressure.

TEMP, TBOT, TE2, TE3, . . ., TTOP
Temperature.

ENKE
Turbulent Kinetic Energy.

ENDS
Energy Dissipation Rate.

VOLT
Electric scalar potential.

SP01SP06
Multiple Species Mass Fraction.

AX
Magnetic vector potential in X direction.

AY
Magnetic vector potential in Y direction.

AZ
Magnetic vector potential in Z direction.

Notes

Deletes the degree of freedom constraints (and all corresponding finite element constraints) on a line previously specified with the **DL** command. See the **DDELE** command for delete details.

Warning: On previously meshed lines, *all* constraints on affected nodes will also be deleted, whether or not they were specified by the DL command.

If the multiple species labels have been changed to user-defined labels via the **MSSPEC** command, use the user-defined labels.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PerfEC>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Displacement>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Species>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Turbulence>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Velocity>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Lines
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>On Lines
Main Menu>Solution>Constraints>Delete>On Lines
Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All Lines
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PerfEC>On Lines
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Lines
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Lines
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Displacement>On Lines
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Pressure DOF>On Lines
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Species>On Lines
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Turbulence>On Lines
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Velocity>On Lines
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Lines
Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Lines
Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Lines

DLIST, *NODE1*, *NODE2*, *NINC*

Lists DOF constraints.

SOLUTION: FE Constraints

MP ME ST <> <> PR EM <> FL PP ED

NODE1, *NODE2*, *NINC*

List constraints for nodes *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If ALL (default), *NODE2* and *NINC* are ignored and constraints for all selected nodes [**NSEL**] are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1*(*NODE2* and *NINC* are ignored).

Notes

Listing applies to the selected nodes [**NSEL**] and the selected degree of freedom labels [**DOFSEL**].

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>DOF Constraints>On All Nodes
Utility Menu>List>Loads>DOF Constraints>On Picked Nodes

DLLIST, *LINE*

Lists DOF constraints on a line.

SOLUTION: Solid Constraints

MP ME ST DY <> PR EM <> <> PP ED

LINE

List constraints for this line. If ALL (default), list for all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*.

Notes

Lists the degree of freedom constraints on a line previously specified with the **DL** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>DOF Constraints>On All Lines
Utility Menu>List>Loads>DOF Constraints>On Picked Lines

DMOVE, *NODE1*, *NODE2*, *NINC*

Digitizes nodes on surfaces and along intersections.

PREP7: Digitizing

MP ME ST DY <> PR EM <> FL PP ED

NODE1, *NODE2*, *NINC*

Digitize nodes *NODE1* through *NODE2* in steps of *NINC*. *NODE2* defaults to *NODE1* and *NINC* defaults to 1.

Notes

Digitizes nodes on undefined surfaces, warped surfaces, and along intersection lines. Two orthogonal views showing the nodes on a plane in each view are required. No surfaces need be specified. Two coordinates are determined from the second view and the other coordinate is retained from the first view. Use the **DIG** command to first define nodes in one view (as determined from the **DSET** command). Then reset the view and use this command to move the nodes to the proper location.

Menu Paths

Main Menu>Preprocessor>Create>Nodes>Digitize Nodes>2-View Digitize

---

DMPEXT, *SMODE*, *TMODE*, *Dmpname*, *Freqb*, *Freqe*, *NSTEPS*

Extracts modal damping coefficients in a specified frequency range.

SOLUTION: Analysis Options

MP ME <> <> <> <> <> <> <> PP ED

SMODE

Source mode number. There is no default for this field; you must enter an integer greater than zero.

TMODE

Target mode. Defaults to *SMODE*.

Dmpname

Array parameter name containing the damping results. Defaults to **d_damp**.

Freqb

Beginning frequency range (real number greater than zero) or EIG at eigenfrequency of source mode. EIG is valid only if *SMODE* = *TMODE*. There is no default for this field; you must enter a value.

Freqe

End of frequency range. Must be blank for *Freqb* = EIG. Default is *Freqb*.

NSTEPS

Number of substeps. Defaults to 1.

Notes

DMPEXT invokes an ANSYS macro that uses modal projection techniques to compute the damping force by the modal velocity of the source mode onto the target mode. From the damping force, damping parameters are extracted. **DMPEXT** creates an array parameter *Dmpname*, with the following entries in each row:

- response frequency

- modal damping coefficient
- modal squeeze stiffness coefficient
- damping ratio
- squeeze-to-structural stiffness ratio

The macro requires the modal displacements from the file **Jobname.EFL** obtained from the **RMFLVEC** command. In addition, a node component FLUN must exist from all FLUID136 nodes. The computed damping ratio may be used to specify constant or modal damping by means of the **DMPRAT** or **MDAMP** commands. For Rayleigh damping, use the **ABEXTRACT** command to compute ALPHAD and BETAD damping parameters. See Chapter 16, " Thin Film Analysis" for more information on thin film analyses.

The macro uses the **LSSOLVE** command to perform two load steps for each frequency. The first load case contains the solution of the source mode excitation and can be used for further postprocessing. Solid model boundary conditions are deleted from the model. In addition, prescribed nodal boundary conditions are applied to the model. You should carefully check the boundary conditions of your model prior to executing a subsequent analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>ThinFilm>DampExtract>Eigenfrequency
Main Menu>Solution>ThinFilm>DampExtract>Frequency Range

DMPRAT, *RATIO*

Sets a constant damping ratio.

SOLUTION: Dynamic Options

MP ME ST DY <> PR <> <> <> PP ED

RATIO
Damping ratio (for example, 2% is input as 0.02).

Command Default

Use damping as defined in the *ANSYS Structural Analysis Guide*.

Notes

Sets a constant damping ratio for use in the harmonic response (**ANTYPE**,HARMIC) analysis (full, reduced, and modal superposition), the mode superposition transient (**ANTYPE**,TRANS) analysis, and the spectrum (**ANTYPE**,SPECTR) analysis.

Note that for structures with multiple materials, **MP**,DMPR can be used to specify constant material damping coefficients for full and modal harmonic analyses. **MP**,DMPR is not applicable for transient or spectrum analyses.

In a full and reduced harmonic analysis, beta damping (**BETAD**) for this command is calculated via Equation 15–23.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping
Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping

DNSOL, *NODE*, *Item*, *Comp*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

Defines or modifies solution results at a node.

POST1: Set Up

MP ME ST DY <> PR EM <> FL PP ED

NODE
Node for which results are specified. If ALL, apply to all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Item
Label identifying results, see DNSOL - Valid Item and Component Labels. Items also require a component label.

Comp
Component of the item. Valid component labels are shown DNSOL - Valid Item and Component Labels below.

V1, *V2*, *V3*, *V4*, *V5*, *V6*
Value assigned to result. If zero, a zero value will be assigned. If blank, the value remains unchanged. Additional values (if any) assigned to the remaining components (in the order corresponding to the *Comp* list shown below for the specified *Item* (starting from the specified *Comp* label and proceeding to the right).

Notes

DNSOL can be used only with FULL graphics activated (**/GRAPHICS**,FULL); it will not work correctly with PowerGraphics activated.

DNSOL defines or modifies solution results in the database at a node. For example, **DNSOL**,35,U,X,.001,.002,.001 assigns values 0.001, 0.002, and 0.001 to UX, UY, and UZ (respectively) for node 35. All results that are changed in the database, including the nodal degree of freedom results, are available for all subsequent operations. All data is stored in the solution coordinate system, but will be displayed in the results coordinate system [**RSYS**]. Use the **PRNSOL** command to list the current results.

Data input by **DNSOL** is stored in temporary space and does not replace information in the database. Therefore, data input by this command may be overwritten if a change is made to the selected set of nodes.

Issuing the **DNSOL** command or its GUI equivalent requires you to place the data type (stress/strain) in the element nodal records. To get around this requirement, use the **DESOL** command or equivalent path to add a "dummy" element stress/strain record.

Result items are available depending on element type; check the individual element for availability. Valid item and component labels for element results are:

DNSOL - Valid Item and Component Labels

Valid Item and Component Labels for Nodal DOF Results

Item	Comp	Description
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.
TEMP[1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity.
A	X, Y, Z	X, Y, or Z magnetic vector potential.
ENKE		Turbulent kinetic energy.
ENDS		Turbulent energy dissipation.

Valid Item and Component Labels for Area and Volume Element Results

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.

Valid Item and Component Labels for Area and Volume Element Results

Item	Comp	Description
"	PLWK	Plastic work/volume.
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component magnetic force or vector sum.

Valid Item Labels for FLOTRAN Nodal Results

Item	Description
TTOT	Total temperature.
HFLU	Heat flux.
HFLM	Heat transfer (film) coefficient.
COND	Fluid laminar conductivity.
PCOE	Pressure coefficient.
PTOT	Total (stagnation) pressure.
MACH	Mach number.
STRM	Stream function. (2-D applications only.)
DENS	Fluid density.
VISC	Fluid laminar viscosity.
EVIS	Fluid effective viscosity.
CMUV	Turbulent viscosity coefficient.
ECON	Fluid effective conductivity.
YPLU	Y+, a turbulent law of the wall parameter.
TAUW	Shear stress at the wall.

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, ..., TTOP instead of TEMP.

Menu Paths

Main Menu>General Postproc>Define/Modify>Nodal Results

DOF, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*, *Lab10*
Adds degrees of freedom to the current DOF set.

PREP7: Element Type

MP ME ST DY <> PR EM <> FL PP ED

Lab1, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*, *Lab10*
Valid labels are: UX, UY, UZ (structural displacements); ROTX, ROTY, ROTZ (structural rotations); TEMP, TBOT, TE2, TE3, . . ., TTOP (temperatures); PRES (pressure); VOLT (voltage); MAG (magnetic scalar potential); AX, AY, AZ (magnetic vector potentials); CURR (current); EMF (electromotive force drop); DELETE.

Command Default

Use degree of freedom set determined from element types.

Notes

The degree of freedom (DOF) set for the model is determined from all element types defined. For example, if only LINK1 is defined, the set is UX,UY. If LINK1 and BEAM3 are defined, the set is UX,UY,ROTZ. This command may be used to add to the current set. The ALL label may be used on some commands to represent all labels of the current degree of freedom set for the model. Issue the **DOF** command with no arguments to list the current set. Use the DELETE label to delete any previously added DOFs and return to the default DOF set.

Menu Paths

Main Menu>Preprocessor>Element Type>Add DOF
Main Menu>Preprocessor>Element Type>Remove DOFs

DOFSEL, *Type*, *Dof1*, *Dof2*, *Dof3*, *Dof4*, *Dof5*, *Dof6*
Selects a DOF label set for reference by other commands.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of select:

S
Select a new set of labels.

A
Add labels to the current set.

U
Unselect (remove) labels from the current set.

ALL
Restore the full set of labels.

STAT
Display the current select status.

Dof1, *Dof2*, *Dof3*, *Dof4*, *Dof5*, *Dof6*
Used only with *Type* = S, A, or U. Valid structural labels: UX, UY, or UZ (displacements); U (UX, UY, and UZ); ROTX, ROTY, or ROTZ (rotations); ROT (ROTX, ROTY, and ROTZ); DISP (U and ROT). Valid thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Valid fluid flow labels: PRES (pressure); VX, VY, or VZ (fluid velocities); V (VX, VY, and VZ); ENKE, ENDS (turbulent kinetic energy, turbulent energy dissipation); EN (ENKE and ENDS turbulent energies) (FLOTRAN). Valid electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current). Valid magnetic labels: MAG (scalar magnetic potential); AX, AY or AZ (vector magnetic potentials); A (AX, AY and AZ); CURR (current). Valid structural force labels: FX, FY, or FZ (forces); F (FX, FY, and FZ); MX, MY, or MZ (moments); M (MX, MY, and MZ); FORC (F and M). Valid thermal force labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow). Valid fluid flow force labels: FLOW (fluid flow). Valid electric force labels: AMPS (current flow), CHRG (electric charge). Valid magnetic force labels: FLUX (scalar magnetic flux); CSGX, CSGY, or CSGZ (magnetic current segments); CSG (CSGX, CSGY, and CSGZ).

Command Default

Degree of freedom (and the corresponding force) labels are determined from the model.

Notes

Selects a degree of freedom label set for reference by other commands. The label set is used on certain commands where ALL is either input in the degree of freedom label field or implied. The active label set has no effect on the solution degrees of freedom. Specified labels which are not active in the model (from the **ET** or **DOF** command) are ignored. As a convenience, a set of force labels corresponding to the degree of freedom labels is also selected. For example, selecting UX also causes FX to be selected (and vice versa). The force label set is used on certain commands where ALL is input in the force label field.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Constraints
Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Forces
Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Constraints
Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Forces
Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Constraints
Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Forces
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Constraints
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Forces

---

DOMEGA, *DOMGX*, *DOMGY*, *DOMGZ*

Specifies the rotational acceleration of the structure.

SOLUTION: Inertia

MP ME ST <> <> PR <> <> <> PP ED

DOMGX, *DOMGY*, *DOMGZ*
Rotational acceleration of the structure about the global Cartesian X , Y, and Z axes.

Notes

Specifies the rotational acceleration of the structure about each of the global Cartesian axes. Rotational accelerations may be defined in analysis types **ANTYPE**,STATIC, HARMIC (full or mode superposition), TRANS (full or mode superposition), and SUBSTR. See the *ANSYS, Inc. Theory Reference* for details. Units are radians/time^2. Related commands are **ACEL**, **CGLOC**, **CGOMGA**, **DCGOMG**, and **OMEGA**.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>Global
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Accel>Global
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>Global
Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Accel>Global

---

DSCALE, *RFACT*, *IFACT*, *TBASE*

Scales DOF constraint values.

SOLUTION: FE Constraints

MP ME ST <> <> PR EM <> FL PP ED

RFACT
Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

IFACT
Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

TBASE
Base temperature for temperature difference. For temperatures, the scale factor is applied to the temperature difference (*T*-*TBASE*) and then added to *TBASE*. *T* is the current temperature.

Notes

Scales degree of freedom constraint values (displacement, temperature, etc.) in the database. Scaling applies to the previously defined values for the selected nodes [**NSEL**] and the selected degree of freedom labels [**DOFSEL**]. Issue **DLIST** command to review results. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

Note — Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

DSCALE does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Constraints
Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Constraints

---

/DSCALE, *WN*, *DMULT*

Sets the displacement multiplier for displacement displays.

GRAPHICS: Scaling

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

DMULT

AUTO or 0
Scale displacements automatically so that maximum displacement (vector amplitude) displays as 5 percent of the maximum model length, as measured in the global Cartesian X, Y, or Z directions. This is the default setting when **NLGEOM** is OFF.

1
Do not scale displacements (i.e., scale displacements by 1.0, true to geometry). Often used with large deflection results. This is the default setting when **NLGEOM** is ON.

FACTOR
Scale displacements by numerical value input for FACTOR.

OFF
Remove displacement scaling (i.e., scale displacements by 0.0, no distortion).

USER
Set *DMULT* to that used for last display (useful when last *DMULT* value was automatically calculated).

Command Default

The default value is 1.0 when **NLGEOM** is ON, and AUTO when **NLGEOM** is OFF.

Notes

If Multi-Plots are not being displayed, and the current device is a 3-D device [**/SHOW**,3D], then the displacement scale in all active windows will be the same, even if separate **/DSCALE** commands are issued for each active window. For efficiency, ANSYS 3-D graphics logic maintains a single data structure (segment), which contains only one displacement scale. The program displays the same segment (displacement scale) in all windows. Only the view settings will be different in each of the active windows.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Displacement Scaling

DSET, *NODE1*, *NODE2*, *NODE3*, *DDEV*

Sets the scale and drawing plane orientation for a digitizing tablet.

PREP7: Digitizing

MP ME ST DY <> PR EM <> FL PP ED

NODE1, *NODE2*, *NODE3*

Any three (noncolinear) nodes defining a plane parallel to the drawing. Nodes and actual locations (in any coordinate system) must have been previously defined.

DDEV

Digitizing device type number (device dependent).

Notes

Sets drawing scale size and defines the drawing plane orientation for use with a digitizing tablet. Drawings must be to scale. Views must represent standard orthogonal parallel projections. The three nodes indicated must be digitized [**DIG**] from the tablet after this command is issued.

Menu Paths

Main Menu>Preprocessor>Create>Nodes>Digitize Nodes>Set Plane/Device

---

DSOPT, *Configopt*, *Ndomains*, *Max_Iter*, *Lev_Diff*

Specify the distributed solver settings.

SOLUTION: Analysis Options

MP ME ST DY <> PR EM <> FL PP ED

Configopt

Selects the distributed mode for solvers.

Local

Sends domains to the local system processors only. The number of processors is set by the **DSPROC** command. This option works for shared memory machines where one machine contains more than one CPU. **EQSLV**,DPCG (or DJCG or DDS) controls the selection of the appropriate executable implicitly.

File

Uses the **config81.dds** file to specify multiprocessor options. This option works for both shared memory and distributed memory machines across a network. The specified executable in the **config81.dds** file should be consistent with the **EQSLV** command setting.

Script

Uses the **ansdds81** script (**ansdds.bat** on Windows platforms) to specify multiprocessor commands and options. This option works for both shared memory and distributed memory machines across a network. The specified executable in the **ansdds81** script should be consistent with the **EQSLV** command.

Ndomains

AUTO

Automatically sets the number of preconditioner domains (default; recommended).

N

Sets the number of preconditioner domains to *N*>1. Typically, the DDS solver uses about 3000 DOFs per preconditioner domain. The DPCG solver uses about 80 elements per domain. For the DPCG solver, the more domains used, the more robust the preconditioner will be, but the more expensive (more memory and CPU time) as well.

Max_Iter

Maximum number of iterations allowed for the distributed solver (default = 300). Most solutions should converge in less than 300 iterations. If the matrix system is ill-conditioned, more than 300 iterations may help achieve convergence. This field applies only to the DDS solver. For DPCG or DJCG, use **EQSLV** to specify the maximum number of iterations.

Lev_Diff

Level of difficulty of the analysis (1 - 4; default = 1). The higher the level of difficulty, the more difficult the analysis the solver can handle and the more memory and CPU time the solver will take. *Lev_Diff* = 1 or 2 is recommended for less difficult problems (such as well-shaped 3-D solid elements). *Lev_Diff* = 3 or 4 is recommended for more ill-conditioned (difficult) problems (more than 30% of elements are beams or shells). For models using nonuniform materials, use a higher level of difficulty. Analyses with *Lev_Diff* = 3 or 4 will converge faster, but with high memory consumption and increased CPU time per iteration.

Notes

DSOPT lets you define machine and network information for running large models on multiple processors using the distributed solvers. The processors may be on different systems, but must be on the same type of platform (all SGI/IRIX systems, for example).

The **DSOPT** command is valid for the DPCG, DJCG, and DDS solvers.

The distributed solvers are part of Parallel Performance for ANSYS, which is a separately-licensed product. See Improving ANSYS Performance and Parallel Performance for ANSYS in the *ANSYS Advanced Analysis Techniques Guide* for more information.

Issue **DSOPT**,STAT to display the current settings.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Sol'n Options
Main Menu>Solution>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Sol'n Controls>Sol'n Options

Note — This menu path is only available if ANSYS is invoked with the -pp command line option.

DSPROC, *NPROC*

Specifies number of processors for a distributed solution.

SOLUTION: Analysis Options

MP ME ST DY <> PR EM EH FL PP <>

NPROC

The number of processors (system dependent) to use. Defaults to 1. For shared memory machines where one machine contains more than one CPU, ANSYS recommends setting *NPROC* no higher than the number of available processors *minus one*; for example, on a four-processor system, set *NPROC* to 3.

For the DDS solver, *NPROC* is used in conjunction with **DSOPT**,local option only. However, for the DPCG or DJCG solvers, you must use this option explicitly. For these cases, *NPROC* is the total number of processors to be used for solving, summed from the entire network. This requirement applies to **DSOPT**,local, file, or script options with the DPCG or DJCG solvers.

Notes

Setting *NPROC* to greater than the number of actual physical processors available may negatively impact performance and is not recommended.

Issue **DSPROC**,STAT to display the current settings.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Sol'n Options
Main Menu>Solution>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Sol'n Controls>Sol'n Options

Note — This menu path is only available if ANSYS is invoked with the -pp command line option.

DSUM, *SIGNIF*, *Label*, *TD*

Specifies the double sum mode combination method.

SOLUTION: Spectrum Options

MP ME ST <> <> PR <> <> <> PP ED

SIGNIF

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single point, multi-point, or DDAM response (**SPOPT**, SPRS, MPRS, or DDAM), the significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT**, PSD.)

Label

Label identifying the combined mode solution output.

DISP
Displacement solution (default). Displacements, stresses, forces, etc., are available.

VELO
Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

ACEL
Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

TD
Time duration for earthquake or shock spectrum. *TD* defaults to 10.

Notes

This command is also valid for PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine

---

DSURF, *KCN*, *XSURF*, *YSURF*, *ZSURF*

Defines the surface upon which digitized nodes lie.

PREP7: Digitizing

MP ME ST DY <> PR EM <> FL PP ED

KCN
Surface is located in coordinate system *KCN*. *KCN* may be 0,1,2 or any previously defined local coordinate system number.

XSURF, *YSURF*, *ZSURF*
Input one value to define the surface constant. Input 999 in the other two fields. Interpret fields as R, θ, Z for cylindrical or R, θ,Φ for spherical or toroidal coordinate systems. *XSURF* and *YSURF* default to 999 if *KCN* = 0.

Command Default

Surface associated with **DIG** command is the global Cartesian X-Y plane with Z = 0.

Notes

Defines the surface upon which the nodes to be digitized (with the **DIG** command) actually lie. Surfaces are defined by a coordinate system number and a coordinate constant [**MOVE**]. Two coordinates are determined from the drawing and converted to surface coordinates. The third coordinate is defined from the input surface constant. If nodes lie on warped or undefined surfaces, use the **DMOVE** command.

Menu Paths

Main Menu>Preprocessor>Create>Nodes>Digitize Nodes>Define Surface

DSYM, *Lab*, *Normal*, *KCN*

Specifies symmetry or antisymmetry DOF constraints on nodes.

SOLUTION: FE Constraints

MP ME ST DY <> PR EM <> <> PP ED

Lab
Symmetry label:

SYMM
Generate symmetry constraints as described below (default).

ASYM
Generate antisymmetry constraints as described below.

Normal
Surface orientation label to determine the constraint set (surface is assumed to be perpendicular to this coordinate direction in coordinate system *KCN*):

X
Surface is normal to coordinate X direction (default). Interpreted as R direction for non-Cartesian coordinate systems.

Y
Surface is normal to coordinate Y direction. θ direction for non-Cartesian coordinate systems.

Z
Surface is normal to coordinate Z direction. Φ direction for spherical or toroidal coordinate systems.

KCN
Reference number of global or local coordinate system used to define surface orientation.

Notes

Specifies symmetry or antisymmetry degree of freedom constraints on the selected nodes. The nodes are first automatically rotated (any previously defined rotations on these nodes are redefined) into coordinate system *KCN*, then zero-valued constraints are generated, as described below, on the selected degree of freedom set (limited to displacement, velocity, and magnetic degrees of freedom) [**DOFSEL**]. Constraints are defined in the (rotated) nodal coordinate system, as usual. See the **D** and **NROTAT** commands for additional details about constraints and nodal rotations.

This command is also valid in PREP7.

Symmetry and Antisymmetry Constraints:

Symmetry or antisymmetry constraint generations are based upon the valid degrees of freedom in the model, i.e., the degrees of freedom associated with the elements attached to the nodes. The degree of freedom labels used in the generation depend on the *Normal* label.

For displacement degrees of freedom, the constraints generated are:

	SYMM		ASYM	
Normal	**2-D**	**3-D**	**2-D**	**3-D**
X	UX, ROTZ	UX, ROTZ, ROTY	UY	UY, UZ, ROTX
Y	UY, ROTZ	UY, ROTZ, ROTX	UX	UX, UZ, ROTY
Z	--	UZ, ROTX, ROTY	--	UX, UY, ROTZ

For velocity degrees of freedom, the constraints generated are:

	SYMM		ASYM	
Normal	**2-D**	**3-D**	**2-D**	**3-D**
X	VX	VX	VY	VY, VZ
Y	VY	VY	VX	VX, VZ
Z	--	VZ	--	VX, VY

For magnetic degrees of freedom, the SYMM label generates flux normal conditions (flux flows normal to the surface). Where no constraints are generated, the flux normal condition is "naturally" satisfied. The ASYM label generates flux parallel conditions (flux flows parallel to the surface).

	SYMM		ASYM	
Normal	**2-D**	**3-D**	**2-D**	**3-D**
X	--	AX	AZ	AY, AZ
Y	--	AY	AZ	AX, AZ
Z	--	AZ	--	AX, AY

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Nodes
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Nodes
Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Nodes

DSYS, *KCN*

Activates a display coordinate system for geometry listings and plots.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

KCN

Coordinate system reference number. *KCN* may be 0,1,2 or any previously defined local coordinate system number.

Note — If a cylinder is displayed in its cylindrical coordinate system (with a 1,0,0 view), it will be unrolled (developed) into a flat plane (with theta along the Y direction).

Command Default

Global Cartesian (*KCN* = 0) display coordinate system.

Notes

Boundary condition symbols, vector arrows, and element coordinate system triads are not transformed to the display coordinate system. The display system orientation (for the default view) is X horizontal to the right, Y vertical upward, and Z out of the screen (normal).

Line directions and area directions (**/PSYMB**,LDIR and **/PSYMB**,ADIR) are not plotted for DSYS >0.

When you create ANSYS 3-D annotation, the coordinates are stored to the database in the DSYS that was active at the time of creation. Changing the DSYS does not change the annotation coordinate data in the database.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Change Display CS to>Global Cartesian
Utility Menu>WorkPlane>Change Display CS to>Global Cylindrical
Utility Menu>WorkPlane>Change Display CS to>Global Spherical
Utility Menu>WorkPlane>Change Display CS to>Specified Coord Sys

---

DTRAN

Transfers solid model DOF constraints to the finite element model.

SOLUTION: Solid Constraints

MP ME ST <> <> PR EM <> <> PP ED

Notes

Constraints are transferred only from selected solid model entities to selected nodes. The **DTRAN** operation is also done if the **SBCTRAN** command is issued, and is automatically done upon initiation of the solution calculations [**SOLVE**].

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Constraints
Main Menu>Solution>Define Loads>Operate>Transfer to FE>Constraints

---

DUMP, *NSTRT*, *NSTOP*

Dumps the contents of a binary file.

AUX2: Binary Files

MP ME ST DY <> PR EM <> FL PP ED

NSTRT, *NSTOP*

Dump file from record *NSTRT* (defaults to 1) to *NSTOP* (defaults to *NSTRT*). If *NSTRT* = HEAD, dump only record 1 of the file (*NSTOP* and the format specification are ignored). If *NSTRT* = ALL, dump the entire file.

Notes

Dumps the file named on the AUX2 **FILEAUX2** command according the format specified on the **FORM** command.

Menu Paths

Utility Menu>File>List>Binary Files
Utility Menu>List>Files>Binary Files

/DV3D, *Lab*, *Key*

Sets 3-D device option modes.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Lab

Mode label:

ACCU

Allows ANSYS to use the accumulation buffer for OpenGL graphics. Activating this feature will provide faster model rotation when shaded backgrounds are in use. This feature is off by default.

ACTR

Label term to designate the cursor position as the center for automatic dynamic rotational center capability . The subsequent *Key* value (see below) turns this capability on and off. This feature is on by default. (Available for OpenGL displays only)

ANIM

Animation mode. The ANIM option allows you to create animation frames in pixmap mode instead of display list mode. This may improve large model performance, but it eliminates local manipulation while animation is in progress. This feature is on by default.

ANTI

Label term to control Anti-aliasing, a smoothing technique for your graph plots. (see below) The subsequent *Key* value turns this capability on and off. The default for this feature is off. (Available for OpenGL displays only).

DGEN
Local manipulation degenerate mode. You access the DGEN option to set wire-frame local manipulation mode for 3-D devices (device dependent). This feature is off by default.

DLIST
With DLIST, you can specify whether screen updates and redraws will be performed using the ANSYS Display List, or the 3-D device's Display List. DLIST is off by default. When using ANSYS on a network, DLIST should be set "ON."

DELS
You use DELS to suppress contour display screen overwrites when **/NOERASE** is active. This prevents the bleed-through that occurs when you overlay contour plots.

TRIS
Triangle strip mode. Tri-stripping provides faster 3-D display capabilities and is on by default. Some display enhancements, such as texturing, are adversely affected by tri-stripping. You can turn off tri-stripping in order to improve these display functions. Be sure to turn tri-stripping on after the desired output is obtained.

Key

The following key options apply to *Lab* = ACCU:

0
(OFF) The accumulation buffer is not accessed. (default)

1
(ON) Access to the buffer is enabled.

The following key options apply to *Lab* = ACTR:

0
(OFF) The cursor position has no effect on the existing rotational center for dynamic operations.

1
(ON) The rotational center for dynamic rotations in OpenGL is determined by the position of the mouse cursor on (or within 15 pixels of) the model. Any rotations that are initiated with the cursor more than 15 pixels from the model will occur about the midpoint of the Z-axis at that point in space. If the Z-buffer has not been refreshed the Z-axis will have an infinite value, and rotations will appear to occur about an extremely long Z-axis. This behavior stops when the graphics window is refreshed or replotted. (default)

Note that when using the GUI in 3-D mode, when ACTR = 1, the **Rotational Center** option is grayed out under **Utility Menu> PlotCtrls> View Setting** because the rotational center is determined strictly by the position of the mouse cursor.

The following key options apply to *Lab* = ANIM:

0
Display list animation. The object can be dynamically manipulated while animating.

1
On UNIX, device-dependent pixmap animation is used. On the PC, bitmap animation is provided (default).

2
On the PC only, this option provides AVI animation which uses the AVI movie player .

Although you can create animations of multiple ANSYS window schemes, animations created with OpenGL display lists (**/DV3D**, ANIM, 0) do not retain the windowing scheme information. You CAN save

multiple windows via the X11/WIN32 drivers, or via the OpenGL driver with **/DV3D**, ANIM, KEY in effect (where KEY is not zero).

The following key options apply to *Lab* = ANTI

0
(OFF) Anti-aliasing is not active (default).

1
(ON) The anti-aliasing technique will be applied to smooth the lines in your displays (valid for OpenGL only).

The following key options apply to *Lab* = DGEN:

0
Normal manipulation.

1
Wireframe Manipulation.

The following key options apply to *Lab* = DLIST:

0
(OFF) The ANSYS Display List is used for plotting and dynamic graphics manipulation (default).

1
(ON) The local (3-D Device) Display List is used for plotting and dynamic rotation.

The following key options apply to *Lab* = TRIS:

0
(OFF) Tri-stripping is off.

1
(ON) Tri-stripping is on (default).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Device Options

DVMORPH, *VOLU*, *XAREA*, *RMSHKY*

Move nodes in selected volumes to conform to structural displacements.

PREP7: Morphing

MP <> <> <> <> <> <> <> <> PP ED

V0LU
Non-structural volume to which mesh movement (morph) applies. If ALL, apply morphing to all selected volumes [**VSEL**]. If *V0LU* = P, graphical picking is enabled. A component may be substituted for *V0LU*.

XAREA
Areas to be excluded from morphing. If ALL, exclude all selected areas [**ASEL**]. If *XAREA* = P, graphical picking is enabled. A component may be substituted for *XAREA*. If *XAREA* is blank (default), allow morphing of nodes

attached to areas of the selected volumes (*VOLU*) which are not shared by unselected volumes. (See Notes for clarification).

RMSHKY
Remesh flag option:

0
Remesh the selected non-structural volumes only if mesh morphing fails.

1
Remesh the selected non-structural volumes and bypass mesh morphing.

2
Perform mesh morphing only and do not remesh.

Notes

The selected volumes should include only non-structural regions adjacent to structural regions. **DVMORPH** will morph the non-structural volumes to coincide with the deflections of the structural regions.

Nodes in the structural regions move in accordance with computed displacements. Displacements from a structural analysis must be in the database prior to issuing **DVMORPH**.

By default, nodes attached to areas can move along the areas. You can use *XAREA* to restrain nodes on certain areas.

By default (*RMSHKY*=0), **DVMORPH** will remesh the selected non-structural volumes entirely if a satisfactory morphed mesh cannot be provided.

If boundary conditions and loads are applied directly to nodes and elements, the **DVMORPH** command requires that these be removed before remeshing can take place.

Exercise care with initial conditions defined by the **IC** command. Before a structural analysis is performed for a sequentially coupled analysis, the **DVMORPH** command requires that initial conditions be removed from all null element type nodes in the non-structural regions. Use **ICDELE** to delete the initial conditions.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Phys Morphing>Volumes

---

DYNOPT

Specifies "Dynamic analysis options" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Dynamics Options

E Commands

E, *I*, *J*, *K*, *L*, *M*, *N*, *O*, *P*
Defines an element by node connectivity.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

I
Number of node assigned to first nodal position (node I). If *I* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

J, *K*, *L*, *M*, *N*, *O*, *P*
Number assigned to second (node J) through eighth (node P) nodal position, if any.

Notes

Defines an element by its nodes and attribute values. Up to 8 nodes may be specified with the **E** command. If more nodes are needed for the element, use the **EMORE** command. The number of nodes required and the order in which they should be specified are described in Chapter 4 of the *ANSYS Elements Reference* for each element type. Elements are automatically assigned a number [**NUMSTR**] as generated. The current (or default) MAT, TYPE, REAL, SECNUM and ESYS attribute values are also assigned to the element.

When creating elements with more than 8 nodes using this command and the **EMORE** command, it may be necessary to turn off shape checking using the **SHPP** command before issuing this command. If a valid element type can be created without using the additional nodes on the **EMORE** command, this command will create that element. The **EMORE** command will then modify the element to include the additional nodes. If shape checking is active, it will be performed before the **EMORE** command is issued. Therefore, if the shape checking limits are exceeded, element creation may fail before the **EMORE** command modifies the element into an acceptable shape.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Auto Numbered>Thru Nodes

---

EALIVE, *ELEM*
Reactivates an element (for the birth and death capability).

SOLUTION: Birth and Death

MP ME ST <> <> <> <> <> <> PP ED

ELEM
Element to be reactivated. If ALL, reactivate all selected elements [**ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *ELEM*.

Notes

Reactivates the specified element when the birth and death capability is being used. An element can be reactivated only after it has been deactivated with the **EKILL** command. Reactivated elements have a zero strain (or thermal heat storage, etc.) state.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Birth & Death>Activate Elem
Main Menu>Solution>Load Step Opts>Other>Birth & Death>Activate Elem

---

EDADAPT, *PART*, *Key*

Activates adaptive meshing in an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

PART
Part ID (number) for which adaptive meshing is to be turned on (or off). Use *PART* = STAT to list the current adaptive meshing definitions.

Key
Adaptivity key:

OFF
Do not use adaptive meshing for the specified part ID (default).

ON
Use adaptive meshing for the specified part ID.

Command Default

Adaptive meshing is off for all parts in the model.

Notes

When adaptive meshing (adaptivity) is turned on, the mesh will automatically be regenerated to ensure adequate element aspect ratios. Adaptive meshing is most commonly used in the analysis of large deformation processes such as metal forming, in which the blank would need to be adaptively meshed.

Adaptive meshing is only valid for parts consisting of SHELL163 elements. By default, adaptive meshing is OFF for all parts in the model. To specify adaptive meshing for more than one part in the model, you must issue the **EDADAPT** command for each part ID. Use the **EDPART** command to create and list valid part IDs. Use the **EDCADAPT** command to define additional adaptive meshing parameters.

The **EDADAPT** command is not supported in an explicit dynamic full restart analysis (**EDSTART**,3). In addition, a full restart cannot be performed successfully if adaptive meshing was used in the previous analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Analysis Options>Adaptive Meshing>Apply to Part
Main Menu>Solution>Analysis Options>Adaptive Meshing>Status

EDALE, *Option*, --, *AFAC*, *BFAC*, --, *DFAC*, *EFAC*, *START*, *END*

Assigns mesh smoothing to explicit dynamic elements that use the ALE formulation.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed:

ADD
Add smoothing controls (default).

DELETE
Delete smoothing controls.

LIST
List smoothing controls.

--
Unused field.

AFAC
Simple average smoothing weight factor (default = 0).

BFAC
Volume weighted smoothing weight factor (default = 0).

--
Unused field.

DFAC
Equipotential smoothing weight factor (default = 0).

EFAC
Equilibrium smoothing weight factor (default = 0). *EFAC* is only applicable to PLANE162 elements.

START
Start time for ALE smoothing (default = 0).

END
End time for ALE smoothing (default = 1e20).

Command Default

The Lagrangian formulation is used for all elements by default.

Notes

Mesh smoothing specified by the **EDALE** command is only applicable to PLANE162 and SOLID164 elements that are flagged to use the ALE formulation (KEYPOPT(5) = 1). To activate the ALE formulation, you must specify at

least one smoothing weight factor on this command and the number of cycles between advection (*NADV*) on the **EDGCALE** command. See Arbitrary Lagrangian-Eulerian Formulation in the *ANSYS LS-DYNA User's Guide* for more information.

The **EDALE** command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Analysis Options>ALE Options>Define
Main Menu>Solution>Analysis Options>ALE Options>Delete All
Main Menu>Solution>Analysis Options>ALE Options>List All

EDASMP, *Option*, *ASMID*, *PART1*, *PART2*, *PART3*, *PART4*, *PART5*, *PART6*, *PART7*, *PART8*, *PART9*, *PART10*, *PART11*, *PART12*, *PART13*, *PART14*, *PART15*, *PART16*
Creates a part assembly to be used in an explicit dynamic analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the part assembly option to be performed.

ADD
Adds a part assembly (default).

DELETE
Deletes a part assembly.

LIST
Lists each part assembly number, and the part numbers that make up each part assembly.

ASMID
User defined part assembly ID number. The part assembly number cannot be the same as any currently defined part ID number.

PART1, ... PART16
Part numbers to be included in the assembly (up to 16 different parts).

Command Default

Default for *Option* is ADD. You must specify *ASMID* to avoid an error message.

Notes

Several ANSYS LS-DYNA commands (such as **EDCGEN**, **EDPVEL**, and **EDIS**) refer to assembly ID numbers. If you intend to use assembly ID numbers with these commands, you must first define the assembly ID numbers using **EDASMP**.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Assembly Options

EDBOUND, *Option*, *Lab*, *Cname*, *XC*, *YC*, *ZC*, *Cname2*, *COPT*

Defines a boundary plane for sliding or cyclic symmetry.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option

Label identifying the symmetry plane option to be performed.

ADD
: Define a sliding or cyclic symmetry plane.

DELE
: Delete a specified sliding or cyclic symmetry plane.

LIST
: List defined sliding or cyclic symmetry planes.

Lab

Valid boundary options for defining a symmetry plane. A valid label must always be specified for adding, deleting, or listing boundary planes.

SLIDE
: Sliding symmetry plane.

CYCL
: Cyclic symmetry plane.

Cname

Name of existing component [**CM**] to which boundary symmetry is to be applied or deleted. Component must consist of nodes. For *Option* = LIST, a component is not required because all defined symmetry planes are listed for the specified *Lab*. For *Option* = DELE, use *Cname* = ALL to delete all symmetry planes currently defined for the specified *Lab*.

XC, *YC*, *ZC*

X, Y, and Z coordinates of the head of the vector defining normal (*Lab* = SLIDE) or axis of rotation (*Lab* = CYCL). The tail of the vector is at the global origin.

Cname2

Name of existing nodal component [**CM**] for which second cyclic boundary plane is to be applied. Each node in *Cname2* component is constrained to a corresponding node in the first component set. Therefore, component *Cname2* must have the same number of nodes as the *Cname* component. *Cname2* is valid only for *Lab* = CYCL.

COPT

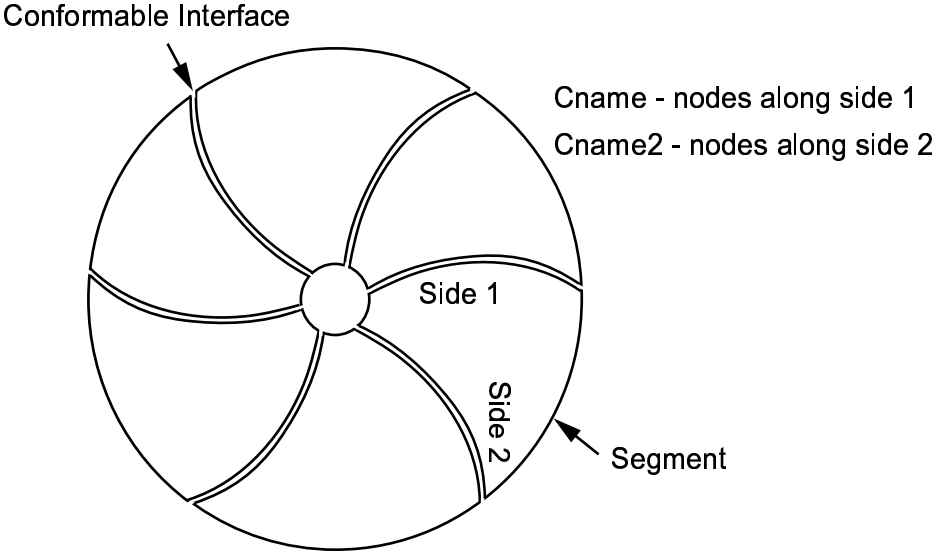
Specified constraint option for sliding plane symmetry. *COPT* is valid only for *Lab* = SLIDE. Valid *COPT* options are:

0
: Nodes move on normal plane (default).

1
: Nodes move only in vector direction.

Notes

For cyclic symmetry, the node numbers in component *Cname2* must differ from the node numbers in *Cname* by a constant offset value. In addition, the nodes in *Cname2* must have locations which, if given in cylindrical coordinates, all differ by the same angle from the nodes in *Cname*. The following figure shows how you would define components for a cyclic symmetry plane.



This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>Symm Bndry Plane
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Symm Bndry Plane>Delete All
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Symm Bndry Plane>Delete Individ
Main Menu>Solution>Constraints>Apply>Symm Bndry Plane
Main Menu>Solution>Constraints>Delete>Symm Bndry Plane>Delete All
Main Menu>Solution>Constraints>Delete>Symm Bndry Plane>Delete Individ

---

EDBX, *Option*, *BOXID*, *XMIN*, *XMAX*,*YMIN*, *YMAX*, *ZMIN*, *ZMAX*

Creates a box shaped volume to be used in a contact definition for explicit dynamics.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the contact box definition option to be performed.

ADD
Adds a contact box definition (default).

DELETE
Deletes a contact box definition.

LIST
Lists each box ID number, and the coordinates that make up each box shaped volume.

BOXID
User defined list ID number.

XMIN
Minimum x-coordinate.

XMAX
Maximum x-coordinate.

YMIN
Minimum y-coordinate.

YMAX
Maximum y-coordinate.

ZMIN
Minimum z-coordinate.

ZMAX
Maximum z-coordinate.

Command Default

Default for *Option* is ADD. You must specify *BOXID* to avoid an error message.

Notes

The ANSYS LS-DYNA command **EDCGEN** allows you to define contact and target volumes using box ID numbers BOXID1 and BOXID2, respectively. If you use these arguments to define contact volumes, you must first define their coordinates using the **EDBX** command.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Contact>Define Box

EDBVIS, *QVCO*, *LVCO*

Specifies global bulk viscosity coefficients for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

QVCO
Quadratic viscosity coefficient (defaults to 1.5).

LVCO
Linear viscosity coefficient (defaults to 0.06).

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Bulk Viscosity

Main Menu>Preprocessor>Material Props>Bulk Viscosity
Main Menu>Solution>Analysis Options>Bulk Viscosity
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Bulk Viscosity

EDCADAPT, *FREQ*, *TOL*, *OPT*, *MAXLVL*, *BTIME*, *DTIME*, *LCID*, *ADPSIZE*, *ADPASS*, *IREFLG*, *ADPENE*, *ADPTH*, *MAXEL*

Specifies adaptive meshing controls for an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

FREQ
Time interval between adaptive mesh refinements (default = 0.0). Use *FREQ* = STAT to list the current adaptive meshing control settings.

TOL
Adaptive angle tolerance (in degrees) for which adaptive meshing will occur (default = 1e31). If the relative angle change between elements exceeds the specified tolerance value, the elements will be refined.

OPT
Adaptivity option:

1
Angle change (in degrees) of elements is based on original mesh configuration (default).

2
Angle change (in degrees) of elements is incrementally based on previously refined mesh.

MAXLVL
Maximum number of mesh refinement levels (default = 3). This parameter controls the number of times an element can be remeshed. Values of 1, 2, 3, 4, etc. allow a maximum of 1, 4, 16, 64, etc. elements, respectively, to be created for each original element.

BTIME
Birth time to begin adaptive meshing (default = 0.0).

DTIME
Death time to end adaptive meshing (default = 1e31).

LCID
Data curve number (previously defined on the **EDCURVE** command) identifying the interval of remeshing (no default). The abscissa of the data curve is time, and the ordinate is the varied adaptive time interval. If LCID is nonzero, the adaptive frequency (*FREQ*) is replaced by this load curve. Note that a nonzero *FREQ* value is still required to initiate the first adaptive loop.

ADPSIZE
Minimum element size to be adapted based on element edge length (default = 0.0).

ADPASS
One or two pass adaptivity option.

0
Two pass adaptivity (default).

1
One pass adaptivity.

IREFLG
Uniform refinement level flag (no default). Values of 1, 2, 3, etc. allow 4, 16, 64, etc. elements, respectively, to be created uniformly for each original element.

ADPENE
Adaptive mesh flag for starting adaptivity when approaching (positive *ADPENE* value) or penetrating (negative *ADPENE* value) the tooling surface (default = 0.0).

ADPTH
Absolute shell thickness level below which adaptivity should begin. This option works only if the adaptive angle tolerance (*TOL*) is nonzero. If thickness based adaptive remeshing is desired without angle change, set *TOL* to a large angle. The default is *ADPTH* = 0.0, which means this option is not used.

MAXEL
Maximum number of elements at which adaptivity will be terminated (no default). Adaptivity is stopped if this number of elements is exceeded.

Command Default

No adaptive meshing.

Notes

The **EDCADAPT** command globally sets the control options for all part IDs that are to be adaptively meshed (see the **EDADAPT** command). Because *FREQ* defaults to zero, you must input a nonzero value in this field in order to activate adaptive meshing. You must also specify a reasonable value for *TOL* since the default adaptive angle tolerance (1e31) will not allow adaptive meshing to occur.

The **EDCADAPT** command is not supported in an explicit dynamic full restart analysis (**EDSTART**,3).

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Analysis Options>Adaptive Meshing>Global Settings
Main Menu>Solution>Analysis Options>Adaptive Meshing>Status

EDCGEN, *Option*, *Cont*, *Targ*, *FS*, *FD*, *DC*, *VC*, *VDC*, *V1*, *V2*, *V3*, *V4*, *BTIME*, *DTIME*, *BOXID1*, *BOXID2*

Specifies contact parameters for an explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the contact behavior (dictates the meaning of *V1* through *V4*).

AG
Automatic general contact.

ANTS
Automatic nodes-to-surface contact.

ASSC
Automatic single surface contact.

ASS2D
Automatic 2-D single surface contact (applicable only with the *Cont* and *FS* arguments).

ASTS
Automatic surface-to-surface contact.

DRAWBEAD
Drawbead contact

ENTS
Eroding nodes-to-surface contact.

ESS
Eroding single surface contact.

ESTS
Eroding surface-to-surface contact.

FNTS
Forming nodes-to-surface contact.

FOSS
Forming one way surface-to-surface contact.

FSTS
Forming surface-to-surface contact.

NTS
Nodes-to-surface contact.

OSTS
One way surface-to-surface contact.

RNTR
Rigid nodes to rigid body contact.

ROTR
Rigid body to rigid body (one way) contact.

SE
Single edge contact.

SS
Single surface contact.

STS
Surface-to-surface contact.

TDNS
Tied nodes-to-surface contact.

TSES
Tied shell edge-to-surface contact.

TDSS
Tied surface-to-surface contact.

TNTS
Tiebreak nodes-to-surface contact

TSTS
Tiebreak surface-to-surface contact.

Cont
Contact surface identified by a component name [**CM**] , a part ID number [**EDPART**], or an assembly ID number [**EDASMP**]. If a component name is input, the component must contain nodes that represent the contact surface (assemblies are not valid for a component name). Alternatively, a part number may be input that identifies a group of elements as the contact surface, or an assembly number may be input containing a maximum of 16 parts. The assembly ID number must be greater than the highest number used for the part ID. *Cont* is not required for automatic general contact, single edge contact, and single surface contact options (*Option* = AG, SE, ASSC, ESS, and SS). If *Option* = ASS2D, only part assemblies are valid.

Targ
Target surface identified by a component name [**CM**] , a part ID number [**EDPART**], or an assembly ID number [**EDASMP**]. If a component name is input, the component must contain nodes that represent the target surface (assemblies are not valid for a component name). Alternatively, a part number may be input that identifies a group of elements as the target surface, or an assembly number may be input containing a maximum of 16 parts. The assembly ID number must be greater than the highest number used for the part ID. *Targ* is not defined for automatic general contact, single edge contact, automatic single surface contact, eroding single surface contact, single surface contact, and automatic 2-D single surface contact options (*Option* = AG, SE, ASSC, ESS, SS, and ASS2D).

FS
Static friction coefficient (defaults to 0).

FD
Dynamic friction coefficient (defaults to 0).

DC
Exponential decay coefficient (defaults to 0).

VC
Coefficient for viscous friction (defaults to 0).

VDC
Viscous damping coefficient in percent of critical damping (defaults to 0).

V1, *V2*, *V3*, *V4*
Additional input for drawbead, eroding, rigid, and tiebreak contact. The meanings of *V1*-*V4* will vary, depending on *Option*. See the table below for *V1*-*V4* definitions.

Additional input for drawbead contact (*Option* = DRAWBEAD):

V1
Load curve ID giving the bending component of the restraining force per unit draw bead length as a function of draw bead displacement. *V1* must be specified.

V2
Load curve ID giving the normal force per unit draw bead length as a function of draw bead displacement. *V2* is optional.

V3
Draw bead depth.

V4
Number of equally spaced integration points along the draw bead (default = 0, in which case ANSYS LS-DYNA calculates this value based on the size of the elements that interact with the draw bead).

Additional input for eroding contact (*Option* = ENTS, ESS, or ESTS):

V1

Symmetry plane option. The purpose of this option is to retain the correct boundary conditions in a model with symmetry.

0

Off (default).

1

Do not include faces with normal boundary constraints (e.g., segments of brick elements on a symmetry plane).

V2

Erosion/interior node option.

0

Erosion occurs only at exterior boundaries.

1

Interior eroding contact can occur (default).

V3

Adjacent material treatment for solid elements.

0

Solid element faces are included only for free boundaries (default).

1

Solid element faces are included if they are on the boundary of the material subset. This option also allows erosion within a body and the consequent treatment of contact.

Additional input for rigid contact (*Option* = RNTR or ROTR):

V1

Data curve id for force versus deflection behavior [**EDCURVE**]. Also specify *V2*. (No default.)

V2

Force calculation method for rigid contact. (No default.)

1

Data curve gives total normal force on surface versus maximum penetration of any node (only applicable for *Option* = ROTR).

2

Data curve gives normal force on each node versus penetration of node through the surface (*Option* = RNTR or ROTR).

3

Data curve gives normal pressure versus penetration of node through the surface (only applicable for *Option* = RNTR).

V3

Unloading stiffness for rigid contact. This should not be larger than the maximum value used in the data curve. The default is to unload along the data curve (specified on *V1*).

Additional input for tiebreak surface-to-surface contact (*Option* = TSTS). *V1* and *V2* are used to calculate the failure criterion:

V1
Normal failure stress. (No default.)

V2
Shear failure stress. (No default.)

Additional input for tiebreak nodes-to-surface contact (*Option* = TNTS). *V1* through *V4* are used to calculate the failure criterion:

V1
Normal failure force. Only tensile failure (i.e., tensile normal forces) will be considered in the failure criterion. (No default.)

V2
Shear failure force. (No default.)

V3
Exponent for normal force. (Defaults to 2.)

V4
Exponent for shear force. (Defaults to 2.)

BTIME
Birth time for which contact definition will become active (defaults to 0.0).

DTIME
Death time for which contact definition will become inactive (defaults to 1e21).

BOXID1
Contact volume as defined using the **EDBX** command (valid only when defining contact with parts or assemblies).

BOXID2
Target volume as defined using the **EDBX** command (valid only when defining contact with parts or assemblies).

Command Default

No contact defined.

Notes

The frictional coefficient used for contact is determined from *FS*, *FD*, and *DC*, and is assumed to be dependent on the relative velocity of the surfaces in contact:

$\mu c = FD+(FS-FD)\ e^{-DC(v_{rel})}$

The coefficient for viscous friction, *VC*, is necessary to limit the friction force to a maximum. A limiting force is computed:

$F_{lim} = VC(A_{cont})$

where A_{cont} is the area of the segment contacted by the node in contact. The suggested value for *VC* is to use the yield stress in shear:

$$VC = \frac{\sigma_o}{\sqrt{3}}$$

where σ_o is the yield stress of the contacted material.

If a part number is input for *Cont* or *Targ*, it must represent a valid explicit dynamics part definition. For example, an elastic material for explicit dynamics requires EX, NUXY, and DENS. If any part of the material definition is missing for the PART, the **EDCGEN** command will be ignored.

In addition to the contact parameters on this command, you can specify penalty scale factors for the contact (slave) and target (master) surfaces by using the **EDCMORE** command.

Duplicate definitions of the same contact type on the same components or parts will cause only one contact to be defined (previous definitions will be ignored). Duplicate definitions of different contact types on the same components or parts will cause multiple contact types to be defined.

Use the **EDCLIST** and **EDDC** commands to list and delete contact surface specifications. Use the **EDPC** command to select and plot contact entities.

The **EDCGEN** command is not supported in an explicit dynamic full restart analysis (**EDSTART**,3). Thus, you cannot add new contact specifications in a full restart. You can use the **EDCLIST** command to list any contact specifications that were defined in the previous analysis.

EDCGEN is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Contact>Define Contact

---

EDCLIST, *NUM*

Lists contact entity specifications in an explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

NUM

Number identifying contact entity to be listed. Use *NUM* = ALL to list all contact entities (ALL is the default).

Notes

Lists contact entity specifications previously defined with the **EDCGEN** command. The listing will include any contact parameters defined using the **EDCMORE** command.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Contact>List Entities

EDCMORE, *Option*, *NUM*, --, *VAL1*,*VAL2*

Specifies additional contact parameters for a given contact definition in an explicit dynamic analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed.

ADD
Define contact parameters for the contact entity specified by *NUM* (default).

DELE
Delete contact parameters (*VAL1* and *VAL2*) for the contact entity specified by *NUM*. If *NUM* = ALL, all contact parameters previously defined by **EDCMORE** are deleted.

NUM
Contact entity number. This contact entity must have been previously defined with the **EDCGEN** command. Use **EDCLIST** to obtain a list of contact entity numbers.

--
Unused field.

VAL1
Penalty scale factor for slave (contact) surface (SFS); default = 1.

VAL2
Penalty scale factor for master (target) surface (SFM); default = 1.

Command Default

For all contract definitions, SFS = 1 and SFM = 1.

Notes

You can use the **EDCMORE** command to specify two additional contact parameters (SFS and SFM) for a specific contact definition. These parameters will apply only to the contact entity number entered on the *NUM* field. Use the **EDCLIST** command to obtain a list of contact definitions and their corresponding contact entity numbers. The listing produced by **EDCLIST** will include any contact parameters specified with the **EDCMORE** command.

When you use the **EDDC** command to delete a contact definition, any parameters you specified with **EDCMORE** for that contact definition will also be deleted. To delete only the parameters specified by **EDCMORE** for a given contact definition, use the command **EDCMORE**,DELE,*NUM*.

Note — When you delete a contact definition with the **EDDC** command, the contact entity numbers will be renumbered for the remaining contact definitions. Therefore, you should always issue **EDCLIST** to obtain a current list of contact entity numbers before adding or deleting contact parameters with the **EDCMORE** command.

The **EDCMORE** command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Contact>Additional Parms

---

EDCNSTR, *Option*, *Ctype*, *Comp1*, *Comp2*, *VAL1*

Defines various types of constraints for an explicit dynamic analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed.

ADD
Define a constraint (default).

DELE
Delete the constraint specified by *Ctype*, *Comp1*, and *Comp2*. If *Ctype* = ALL, all constraints are deleted.

LIST
List all of the constraints previously defined by the **EDCNSTR** command.

Ctype
Constraint type. The command format will vary, depending on the *Ctype* value.

ENS
Extra node set added to an existing rigid body.

NRB
Nodal rigid body.

STS
Tie between a shell edge and solid elements.

RIVET
Massless rivet between two noncoincident nodes.

If *Ctype* = ENS, the command format is **EDCNSTR**,*Option*,ENS,*Comp1*,*Comp2*

Comp1
Part number of the existing rigid body to which nodes will be added. The rigid body must be previously defined with the **EDMP** command. **EDMP** defines a rigid body based on material reference number (MAT). You must determine the corresponding part number (**EDPART**) for input in this field.

Comp2
Component name identifying extra nodes to be added to the rigid body specified by *Comp1*. *Comp2* must be a nodal component and must not be attached to any other rigid body.

If *Ctype* = NRB, the command format is **EDCNSTR**,*Option*,NRB,*Comp1*, --,*VAL1*

Comp1
Component name identifying a set of nodes that are to be defined as a rigid body. The component may consist of nodes from several different deformable parts.

--
This field is not used for *Ctype* = NRB.

VAL1
Coordinate system ID number (CID) to be used for output of data. The coordinate system must have been previously defined with the **EDLCS** command.

If *Ctype* = STS, the command format is **EDCNSTR**,*Option*,STS,*Comp1*,*Comp2*

Comp1
Node number of the shell element node that will be tied to solid element nodes that are specified by *Comp2*. The *Comp1* node must lie along the edge of a shell element and be coincident to at least one node included in *Comp2*.

Comp2
Component name consisting of solid element nodes (up to nine nodes) to which the shell element node will be tied. *Comp2* must consist of nodes that are on solid elements, and the nodes must define a line that will remain linear throughout the analysis. At least one of the nodes in *Comp2* must be coincident with the shell node specified in *Comp1*.

If *Ctype* = RIVET, the command format is **EDCNSTR**,*Option*,RIVET,*Comp1*,*Comp2*

Comp1
Node number of the first node on the rivet.

Comp2
Node number of the second node on the rivet. This node cannot have the same coordinates as the first node specified by *Comp1*.

Notes

The **EDCNSTR** command allows you to define several types of constraints in an explicit dynamic analysis. A brief description of each constraint type is given below. See Constraints and Initial Conditions in the *ANSYS LS-DYNA User's Guide* for more information.

Extra Node Set Added to a Rigid Body (Ctype = ENS)

The ability to add extra nodes to an existing rigid body has many potential applications, including placing nodes where joints will be attached between rigid bodies, defining nodes where point loads will be applied, and defining a lumped mass at a specific location. The extra nodes specified by *Comp2* may be located anywhere in the model and may have coordinates outside those of the original rigid body specified by *Comp1*.

Nodal Rigid Body (Ctype = NRB)

Unlike typical rigid bodies that are defined with the **EDMP** command, nodal rigid bodies defined with the **EDCNSTR** command are not associated with a part number. This can be advantageous for modeling rigid (welded) joints in a model. For a rigid joint, portions of different flexible components (having different MAT IDs) act together as a rigid body. It is difficult to define this type of rigid body with a unique MAT ID (and corresponding part number). However, the rigid joint can be easily defined using a nodal rigid body.

Shell Edge to Solid Tie (Ctype = STS)

The STS option ties regions of solid elements to regions of shell elements. A single shell node may be tied to up to nine brick element nodes that define a "fiber" vector. Solid element nodes constrained in this way remain linear throughout the analysis but can move relative to each other in the fiber direction.

Rivet between Two Nodes (Ctype = RIVET)

The RIVET option defines a massless rigid constraint between two nodes, similar to spotwelds defined with the **EDWELD** command. Unlike a spotweld, however, rivets contain nodes that are noncoincident, and failure cannot be specified. When a rivet is defined, the distance between the nodes is kept constant throughout any motion that occurs during a simulation. Nodes connected by a rivet cannot be part of any other constraints specified in the model.

The **EDCNSTR** command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>Additional Nodal
Main Menu>Solution>Constraints>Apply>Additional Nodal

EDCONTACT, *SFSI*, *RWPN*, *IPCK*, *SHTK*, *PENO*, *STCC*, *ORIE*, *CSPC*, *PENCHK*

Specifies contact surface controls for an explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

SFSI
Scale factor for sliding interface penalties. Defaults to 0.1.

RWPN
Scale factor for rigid wall penalties (defaults to 0). If *RWPN* = 0, rigid bodies interacting with rigid walls are not considered. If *RWPN*>0, rigid bodies interact with fixed rigid walls. A value of 1.0 should be optimal; however, this may be problem dependent.

IPCK
Initial contact surface penetration checking option:

1
No checking.

2
Full check of initial penetration is performed (default).

SHTK
Shell thickness contact option for surface-to-surface and nodes-to-surface contact (see Notes below):

0
Thickness is not considered (default).

1
Thickness is considered, except in rigid bodies.

2
Thickness is considered, including rigid bodies.

PENO
Penalty stiffness option (options 4 and 5 are useful for metal forming calculations):

1
Minimum of master segment and slave node (default).

2
Use master segment stiffness.

3
Use slave node value.

4
Use area or mass weighted slave node value.

5
Use slave node value inversely proportional to shell thickness. (This may require special scaling and is not generally recommended.)

STCC
Shell thickness change option for single surface contact:

1
Shell thickness changes are not considered (default).

2
Shell thickness changes are included.

ORIE
Option for automatic reorientation of contact surface segments during initialization:

1
Activate for automated (part ID) input only (default).

2
Activate for manual (nodal component) and automated (part ID) input.

3
Do not activate.

CSPC
Contact surface penetration check multiplier, used if small penetration checking is on (*PENCHK* = 1 or 2). Defaults to 4.

PENCHK
Small penetration check, used only for contact types STS, NTS, OSTS, TNTS, and TSTS. If the contact surface node penetrates more than the target thickness times *CSPC*, the penetration is ignored and the contacting node is set free. The target thickness is the element thickness for shell elements, or 1/20 of the shortest diagonal for solid elements.

0
Penetration checking is off (default).

1
Penetration checking is on.

2
Penetration checking is on, but shortest diagonal is used.

Notes

The thickness offsets are always included in single surface, automatic surface-to-surface, and automatic nodes-to-surface contact. The shell thickness change option must be used [**EDSHELL**,,,1] and a nonzero value must be specified for *SHTK* before the shell thickness changes can be included in the surface-to-surface contact type. Additionally, *STCC* must be set to 2 if thickness changes are to be included in the single surface contact algorithms.

To reset the contact options to default values, issue the **EDCONTACT** command with no fields specified.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Contact>Advanced Controls

---

EDCPU, *CPUTIME*

Specifies CPU time limit for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

CPUTIME
CPU time limit (in seconds) for the current phase of the analysis (defaults to 0). If *CPUTIME* = 0, no CPU time limit is set. *CPUTIME* values below 0 are not allowed.

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Analysis Options>CPU Limit

EDCRB, *Option*, *NEQN*, *PARTM*, *PARTS*

Constrains two rigid bodies to act as one in an explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed:

ADD
Define an equation to constrain two rigid bodies (default).

DELE
Delete the equation (specified by *NEQN*) that constrains two rigid bodies. If *NEQN* is blank, all equations constraining rigid bodies are deleted.

LIST
List constrained rigid bodies specified by *NEQN*. If *NEQN* is blank, all constrained rigid bodies are listed.

NEQN
Equation reference number. Defaults to *PARTS*. *NEQN* should be a unique number for each pair of *PARTM* and *PARTS*. If it is not unique, the equation reference number defined last will overwrite any previously defined *NEQN* with the same number.

PARTM
PART number [**EDPART**] identifying the master rigid body. This value is ignored if the DELE or LIST labels are specified. No default; you must enter a value.

PARTS
PART number [**EDPART**] identifying the slave rigid body. This value is ignored if the DELE or LIST labels are specified. No default; you must enter a value.

Notes

EDCRB is valid only for materials defined as rigid bodies with the **EDMP**,RIGID command. **EDCRB** automatically generates a constraint equation to force the specified rigid bodies to behave as a single rigid body. The slave rigid body takes on the material properties and loading of the master rigid body. Any loads [**EDLOAD**] existing on the slave rigid body are ignored.

To create a single large rigid body from several smaller bodies, use a series of **EDCRB** commands. With the first command, specify a master and slave to create the first combined rigid body. Then, using that body as the master, specify another slave to create a larger rigid body. Continue the process, using the expanding rigid body as the master and adding slave bodies until you have defined the desired large rigid body. All slave rigid bodies will take on the material properties and loading of the original master rigid body. Note that you will need to use different *NEQN* values for each pair of *PARTM* and *PARTS*. This command will be ignored if you specify the previously-defined master rigid body as a slave rigid body in the same analysis. To change the master and slave definitions, first use the DELE option to delete all master and slave definitions, and then use the ADD option to redefine them.

The equation number, *NEQN*, is a reference number by which the constrained bodies can be identified for listing and deleting purposes on the **EDCRB** command. For any other reference to the constrained bodies (loading, contact definitions, etc.), use the master body part number (*PARTM*).

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Rigid Body CE

EDCSC, *Key*

Specifies whether to use subcycling in an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Key
Subcycling key:

OFF
Do not use subcycling (default).

ON
Use subcycling.

Command Default

No subcycling.

Notes

Subcycling can be used to speed up an analysis when element sizes within a model vary significantly. Relatively small elements will result in a small time step size. When subcycling is on, the minimum time step size is increased for the smallest elements.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Time Controls>Subcycling

EDCTS, *DTMS*, *TSSFAC*

Specifies mass scaling and scale factor of computed time step for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

DTMS
Time step size for mass scaled solutions (defaults to 0).

TSSFAC
Scale factor for computed time step. Defaults to 0.9; if high explosives are used, the default is lowered to 0.67.

Command Default

No mass scaling; scale factor for computed time step = 0.9.

Notes

If *DTMS* is positive, the same time step size will be used for all elements and mass scaling will be done for all elements. Therefore, positive values should only be used if inertial effects are insignificant.

If *DTMS* is negative, mass scaling is applied only to elements whose calculated time step size is smaller than *DTMS*. Negative values should only be used in transient analyses if the mass increases are insignificant.

In order to use mass scaling in an explicit dynamic small restart analysis (**EDSTART**,2) or full restart analysis (**EDSTART**,3), mass scaling must have been active in the original analysis. The time step and scale factor used in the original analysis will be used by default in the restart. You can issue **EDCTS** in the restart analysis to change these settings.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Time Controls>Time Step Ctrls

EDCURVE, *Option*, *LCID*, *Par1*, *Par2*

Specifies data curves for an explicit dynamic analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed.

ADD
Define a data curve (default). If *Option* = ADD, *Par1* and *Par2* must be previously defined array parameters.

DELE
Delete the specified data curve (*LCID*). If *LCID* is blank, all data curves are deleted. *Par1* and *Par2* are ignored for this option.

LIST
List defined data curve (*LCID*). If *LCID* is blank, all data curves are listed. *Par1* and *Par2* are ignored for this option.

PLOT
Plot defined data curve (*LCID*). If *Option* = PLOT, *LCID* must be previously defined with an **EDCURVE** command. Otherwise a warning message will report that *LCID* has not been defined. *Par1* and *Par2* are ignored for this option.

LCID
Data curve ID number (no default). Must be a positive integer.

Par1
Name of user-defined array parameter that contains the abscissa values of the curve data (e.g., time, effective plastic strain, effective strain rate, displacement, etc.).

Par2
Name of user-defined array parameter that contains the ordinate values of the curve data (e.g., damping coefficients, initial yield stress, elastic modulus, force, etc.) corresponding to the abscissa values in *Par1*.

Note — If the length of *Par1* and *Par2* are different, the shortest length will be used.

Notes

EDCURVE can be used to define material data curves (e.g., stress-strain) and load data curves (force-deflection) associated with material models in an explicit dynamics analysis. Material data specified by this command is typically required to define a particular material behavior (e.g., **TB**,HONEY), and the *LCID* number is used as input on the **TBDATA** command.

EDCURVE can also be used to define load curves that represent time dependent loads (force, displacement, velocity, etc.). *Par1* must contain the time values, and *Par2* must contain the corresponding load values. The *LCID* number assigned to the load curve can be used as input on the **EDLOAD** command.

> *Note* — You cannot update a previously defined data curve by changing the array parameters that were input as *Par1* and *Par2*. The data curve definition is written to the database at the time **EDCURVE** is issued. Therefore, subsequent changes to the array parameters that were used as input on **EDCURVE** will not affect the load curve definition. If you need to change the load curve definition, you must delete the load curve (**EDCURVE**,DELE,*LCID*) and define it again.

LCID identifies the data curve. If the value input for *LCID* is the same as the ID number for a data curve previously defined by **EDCURVE**, the previous data will be overwritten. Use **EDCURVE**,LIST and **EDCURVE**,PLOT to check existing data curves.

A starting array element number must be specified for *Par1* and *Par2*. The input for these fields must be a single column array parameter, or a specific column from a multi-column array parameter. When using the GUI with multi-column parameters, you must specify the parameter name and starting position for *Par1* and *Par2* by typing the **EDCURVE** command in the Input Window. This is because only the parameter name is available through the dialog box, which pulls in the first position of a single-column array parameter.

If you need to change a curve definition in an explicit dynamic small restart analysis, issue **EDSTART**,2 first (to specify the restart), then issue the **EDCURVE** command. The revised curve must contain the same number of points as the curve it replaces. This limitation does not apply to a full restart analysis (**EDSTART**,3).

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Curve Options>Add Curve
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Curve Options>Delete Curve
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Curve Options>List Curve
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Curve Options>Plot Curve
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Curve Options>Add Curve
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Curve Options>Delete Curve
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Curve Options>List Curve
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Curve Options>Plot Curve
Main Menu>Preprocessor>Material Props>Curve Options>Add Curve
Main Menu>Preprocessor>Material Props>Curve Options>Delete Curve
Main Menu>Preprocessor>Material Props>Curve Options>List Curve
Main Menu>Preprocessor>Material Props>Curve Options>Plot Curve

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Curve Options>Add Curve
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Curve Options>Delete Curve
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Curve Options>List Curve
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Curve Options>Plot Curve
Main Menu>Solution>Loading Options>Curve Options>Add Curve
Main Menu>Solution>Loading Options>Curve Options>Delete Curve
Main Menu>Solution>Loading Options>Curve Options>List Curve
Main Menu>Solution>Loading Options>Curve Options>Plot Curve

EDDAMP, *PART*, *LCID*, *VALDMP*

Defines mass weighted (Alpha) or stiffness weighted (Beta) damping for an explicit dynamics model.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

PART
PART number [**EDPART**] identifying the group of elements to which damping should be applied. If *PART* = ALL (default), damping is applied to the entire model.

LCID
Load curve ID (previously defined with the **EDCURVE** command) identifying the damping coefficient versus time curve. If time-dependent damping is defined, an *LCID* is required.

VALDMP
Constant system damping coefficient or a scale factor applied to the curve defining damping coefficient versus time.

Notes

Mass-weighted (Alpha) or stiffness-weighted (Beta) damping can be defined with the **EDDAMP** command. Generally, stiffness proportional or beta damping is effective for oscillatory motion at high frequencies. This type of damping is orthogonal to rigid body motion and so will not damp out rigid body motion. On the other hand, mass proportional or alpha damping is more effective for low frequencies and will damp out rigid body motion. The different possibilities are described below:

Global Damping

Mass-weighted or Alpha damping

When *PART* = (blank) or ALL (default), mass-weighted global damping can be defined in the following two ways. In this case, the same damping is applied for the entire structure.

- When the damping coefficient versus time curve (*LCID*) is specified using the **EDCURVE** command, *VALDMP* is ignored by LS-DYNA (although it is written in the LS-DYNA input file **Jobname.K**). The damping force applied to each node in the model is given by $f_d = d(t)mv$, where d(t) is the damping coefficient as a function of time defined by the **EDCURVE** command, m is the mass, and v is the velocity.
- When the *LCID* is 0 or blank (default), a constant mass-weighted system damping coefficient can be specified using *VALDMP*.

The constant and time-dependent damping, described above, cannot be defined simultaneously. The last defined global damping will overwrite any previously defined global damping.

Damping defined for a PART

Mass-weighted or Alpha damping

When *both* a valid *PART* number is specified and the damping coefficient versus time curve (*LCID*) is specified using the **EDCURVE** command, mass-weighted time-dependent damping will be defined for the particular *PART*. In this case, *VALDMP* will act as a scaling factor for the damping versus time curve (if *VALDMP* is not specified, it will default to 1). A valid *PART* number must be specified to define this type of damping. For example, use *PART* =1 (and not blank) when the entire model consists of only one *PART*. Issue the command repeatedly with different *PART* numbers in order to specify alpha damping for different *PARTS*.

Stiffness-weighted or Beta damping

When a valid *PART* number is specified with *LCID* = 0 or (blank) (default), a stiffness-weighted (Beta) constant damping coefficient for this particular *PART* can be defined by *VALDMP*. The stiffness-weighted value corresponds to the percentage of damping in the high frequency domain. For example, 0.1 roughly corresponds to 10% damping in the high frequency domain. Recommended values range from 0.01 to 0.25. Values lower than 0.01 may have little effect. If a value larger than 0.25 is used, it may be necessary to lower the time step size significantly (see the **EDCTS** command). Issue the command repeatedly with different PART numbers in order to specify beta damping for different *PARTS*. Time-dependent stiffness-weighted damping is not available in ANSYS LS-DYNA.

The mass-weighted and stiffness-weighted damping, described above, cannot be defined simultaneously for a particular *PART* number. The last defined damping for the particular PART number will overwrite any previously defined mass-weighted or stiffness-weighted damping for this *PART*.

In order to define the mass-weighted and stiffness-weighted damping simultaneously, you can use the **MP**,DAMP command (instead of the **EDDAMP**,PART, ,VALDMP command) to define stiffness-weighted (Beta) constant damping coefficient. However, do *not* use both of these commands together to define stiffness-weighted (Beta) constant damping coefficient for a particular *PART*. If you do, duplicate stiffness-weighted (Beta) constant damping coefficients for this *PART* will be written to the LS-DYNA input file **Jobname.K**. The last defined value will be used by LS-DYNA. Also, note that the **MP**,DAMP command is applied on the MAT number, and not on the PART number. Since a group of elements having the same MAT ID may belong to more than one *PART* (the opposite is not true), you need to issue the **MP**,DAMP command only once for this MAT ID and the stiffness-weighted (Beta) damping coefficients will be automatically defined for all the *PART*s with that MAT ID.

Mass-weighted and stiffness-weighted damping can be defined simultaneously using the **EDDAMP** command only when mass-weighted damping (constant or time-dependent) is defined as global damping (**EDDAMP**, ALL, LCID, VALDMP) and stiffness-weighted damping is defined for all necessary PARTs (**EDDAMP**,PART, ,VALDMP).

To remove defined *global* damping, reissue the **EDDAMP**, ALL command with *LCID* and *VALDMP* set to 0. To remove damping defined for a particular PART, reissue **EDDAMP**, PART, where PART is the *PART* number, with *LCID* and *VALDMP* set to 0. There is no default for the **EDDAMP** command, i.e., issuing the **EDDAMP** command with PART = LCID = VALDMP = 0 will result in an error. Stiffness-weighted damping defined by the **MP**,DAMP command can be deleted using **MPDELE**, DAMP, MAT.

In an explicit dynamic small restart (**EDSTART**,2) or full restart analysis (**EDSTART**,3), you can only specify global alpha damping. This damping will overwrite any alpha damping input in the original analysis. If you do not input global alpha damping in the restart, the damping properties input in the original analysis will carry over to the restart.

Damping specified by the **EDDAMP** command can be listed, along with other explicit dynamics specifications, by typing the command string **EDSOLV**$**STAT** into the ANSYS input window. Beta damping specified by the **MP**,DAMP command can be listed by **MPLIST**, MAT command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Damping
Main Menu>Preprocessor>Material Props>Damping
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Damping

---

EDDBL, *KEY*

Selects a numerical precision type of the explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

KEY
Number or name identifying numerical precision to be used.

0 or SINGLE
Select single precision version of LS-DYNA (default).

1 or DOUBLE
Select double precision version of LS-DYNA.

STAT
Check the status of the numerical precision in effect.

Notes

Sets the single or double precision version of LS-DYNA into effect. The double precision of LS-DYNA is not available for the HP-UX 10.2, SGI 32-bit, IBM 32-bit, and Linux 32-bit platforms. Please check the availability of the double precision version of LS-DYNA on your system before using the command. If it is not available, use the command default.

The double precision version may be up to 20% slower than the single precision version. The results may also vary based on problem specifications.

In addition to **EDDBL**,STAT, you can use the GUI dialog box to verify which precision version is currently chosen. The GUI is based on the database and is updated to reflect changes.

See Double Precision LS-DYNA for more information.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Solution>Analysis Options>Double Precision

EDDC, *Option*, *Ctype*, *Cont*, *Targ*

Deletes or deactivates/reactivates contact surface specifications in an explicit dynamic analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Option to be performed for contact definition specified by *Ctype*, *Cont*, and *Targ*.

DELE
Delete the specified contact definition (default); valid only in a new analysis.

DACT
Deactivate the specified contact definition; valid only in a small restart.

RACT
Reactivate the specified contact definition (which was previously deactivated); valid only in a small restart.

Ctype
Contact behavior label (see **EDCGEN** command for valid labels).

Cont
Component name or part number [**EDPART**] identifying the contact surface.

Targ
Component name or part number [**EDPART**] identifying the target surface.

Notes

This command allows you to delete or deactivate/reactivate a particular contact specification that was defined by **EDCGEN**. The contact definition is identified by *Ctype*, *Cont*, and *Targ* (Note that *Cont* and *Targ* may not be required for *Ctype* = AG, SE, ASSC, ESS, and SS). The delete option (*Option* = DELE) permanently deletes the contact from the database. Any additional contact parameters defined with the **EDCMORE** command for the contact definition identified on this command will also be deleted or deactivated/reactivated.

You cannot delete contact specifications in an explicit dynamic small restart (**EDSTART**,2). However, you can use *Option* = DACT to deactivate a contact definition that is not needed in the small restart. That contact definition may then be reactivated in a subsequent small restart by using *Option* = RACT.

To delete or deactivate/reactivate all contact specifications for the entire model, use **EDDC**,*Option*,ALL.

The **EDDC** command is not supported in an explicit dynamic full restart analysis (**EDSTART**,3). Thus, you cannot delete, deactivate, or reactivate contact specifications in a full restart that were defined in the previous analysis.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Contact>Activate Entity
Main Menu>Preprocessor>LS-DYNA Options>Contact>Deactvate Entity
Main Menu>Preprocessor>LS-DYNA Options>Contact>Delete Entity

EDDRELAX, *Option*, *NRCYCK*, *DRTOL*, *DFFCTR*, *DRTERM*, *TSSFDR*, *IRELAL*, *EDTTL*

Activates initialization to a prescribed geometry or dynamic relaxation for the explicit analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option

Specifies when dynamic relaxation is activated.

ANSYS

Stresses are initialized in ANSYS LS-DYNA to a prescribed geometry for small strains, according to the solution of an ANSYS (implicit) run. The explicit solution is based on the implicit X,Y,Z displacements and rotations contained in the **drelax** file (created with the **REXPORT** command).

DYNA

Dynamic relaxation is on. When you use this option, you can specify some or all of the parameters *NRCYCK*, *DRTOL*, *DFFCTR*, *DRTERM*, *TSSFDR*, *IRELAL*, and *EDTTL*. Any parameters that you do not specify are set to their default values.

OFF

Turn off initialization to a prescribed geometry (*Option* = ANSYS) or dynamic relaxation (*Option* = DYNA).

NRCYCK

Number of iterations between convergence checks for dynamic relaxation option. Default = 250.

DRTOL

Convergence tolerance for dynamic relaxation option. Default = 0.001.

DFFCTR

Dynamic relaxation factor. Default = 0.995.

DRTERM

Optional termination time for dynamic relaxation. Termination occurs at this time, or when convergence is attained, whichever comes first. Default = infinity.

TSSFDR

Scale factor for computed time step during dynamic relaxation. If zero, the value is set to *TSSFAC* (defined on the **EDCTS** command). After converging, the scale factor is reset to *TSSFAC*.

IRELAL

Automatic control for dynamic relaxation option based on algorithm of Papadrakakis.

0

Not active (default).

1

Active.

EDTTL

Convergence tolerance on automatic control of dynamic relaxation (default = 0.04).

Notes

Use *Option* = ANSYS when running an implicit-to-explicit sequential solution to initialize the structure to a static solution performed earlier by the ANSYS implicit solver. Use *Option* = DYNA to perform dynamic relaxation

within the LS-DYNA program. Use *Option* = OFF to turn off previously specified stress initialization or dynamic relaxation. You must specify the *Option* you want; there is no default.

In LS-DYNA, the dynamic relaxation is performed before the regular transient analysis. The convergence process of the dynamic relaxation is not written to the ANSYS history file. The ANSYS results files only include the converged result of the dynamic relaxation, which is the result at time zero in the **Jobname.HIS** and **Jobname.RST** files.

You can restart a dynamic relaxation analysis (**EDSTART**,2 or **EDSTART**,3) from a previous transient analysis or a previous dynamic relaxation analysis. In the restart, you can change or set the convergence criteria with the **EDDRELAX** command. Only the load curves that are flagged for dynamic relaxation (PHASE = 1 or 2 on **EDLOAD**) are applied after restarting. If you restart the explicit portion of an implicit-to-explicit sequential solution, you do not need to reissue the **REXPORT** command because displacement information contained in the drelax file is already included in the LS-DYNA restart file. If the dynamic relaxation is activated from a regular transient analysis, LS-DYNA continues the output of data to ANSYS results files. This is unlike the dynamic relaxation phase at the beginning of the calculation for which only the converged solution is written.

Menu Paths

Main Menu>Solution>Analysis Options>Dynamic Relax

EDDUMP, *NUM*, *DT*

Specifies output frequency for the explicit dynamic restart file (d3dump).

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

NUM
Number of d3dump (restart) files written during the analysis (defaults to 1). When you specify *NUM*, the time interval between restart files is *TIME* / *NUM*, where *TIME* is the analysis end-time specified on the **TIME** command.

DT
Time interval at which the d3dump (restart) files are written. If *NUM* is input, *DT* is ignored.

Command Default

One restart file is written at the end of the analysis.

Notes

You can use *NUM* or *DT* to specify the time interval at which d3dump restart files will be written. You should not specify both quantities; if both are input, *NUM* will be used. The restart files are written sequentially as d3dump01, d3dump02, etc.

In LS-DYNA, the restart file output is specified in terms of number of time steps. Because the total number of time steps is not known until the LS-DYNA solution finishes, ANSYS calculates an approximate number of time steps for the solution, and then uses *NUM* or *DT* to calculate the required LS-DYNA input. This approximated number of time steps may be different from the total number reached in LS-DYNA after the solution finishes. Therefore, the number of restart dump files or the output interval may differ slightly from what you requested using *NUM* or *DT*.

In an explicit dynamic small restart (**EDSTART**,2) or full restart analysis (**EDSTART**,3), the **EDDUMP** setting will default to the *NUM* or *DT* value used in the original analysis. You can issue **EDDUMP** in the restart to change this setting.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Output Controls>File Output Freq>Number of Steps
Main Menu>Solution>Output Controls>File Output Freq>Time Step Size

---

EDELE, *IEL1*, *IEL2*, *INC*

Deletes selected elements from the model.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

IEL1, *IEL2*, *INC*

Delete elements from *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *INC* (defaults to 1). If *IEL1* = ALL, *IEL2* and *INC* are ignored and all selected elements [**ESEL**] are deleted. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *INC* are ignored).

Notes

Deleted elements are replaced by null or "blank" elements. Null elements are used only for retaining the element numbers so that the element numbering sequence for the rest of the model is not changed by deleting elements. Null elements may be removed (although this is not necessary) with the **NUMCMP** command. If related element data (pressures, etc.) are also to be deleted, delete that data before deleting the elements. **EDELE** is for unattached elements only. You can use the **xCLEAR** family of commands to remove any attached elements from the database.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements
Main Menu>Preprocessor>Modeling>Delete>Elements
Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts

EDENERGY, *HGEN*, *SWEN*, *SIEN*, *RLEN*

Specifies energy dissipation controls for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

HGEN

Hourglass energy control key:

OFF or 0

Hourglass energy is not computed.

ON or 1
Hourglass energy is computed and included in the energy balance (default).

SWEN
Stonewall energy dissipation control key:

OFF or 0
Stonewall energy dissipation is not computed.

ON or 1
Stonewall energy dissipation is computed and included in the energy balance (default).

SIEN
Sliding interface energy dissipation control key:

OFF or 0
Sliding interface energy dissipation is not computed.

ON or 1
Sliding interface energy dissipation is computed and included in the energy balance (default).

RLEN
Rayleigh (damping) energy dissipation control key:

OFF or 0
Rayleigh energy dissipation is not computed.

ON or 1
Rayleigh energy dissipation is computed and included in the energy balance (default).

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Analysis Options>Energy Options

EDFPLOT, *Key*

Allows plotting of explicit dynamics forces and other load symbols.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Key
Load symbol plotting key.

ON or 1
Turn display of load symbols on (default).

OFF or 0
Turn display of load symbols off.

Command Default

Display of load symbols on.

Notes

You must issue **EDFPLOT**,ON to display explicit dynamics load symbols. The explicit load symbols are erased automatically upon a subsequent plot command.

An explicit load symbol always indicates a positive load direction (e.g., positive X direction for FX load), even if the load value is negative. The load symbol does not reflect the load magnitude. You can use standard ANSYS symbol controls to control the appearance of the load symbol. No load symbol is displayed for temperature loads.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Show Forces
Main Menu>Solution>Loading Options>Show Forces
Utility Menu>PlotCtrls>Symbols

EDGCALE, *NADV*, *METH*

Defines global ALE controls for an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

NADV
Number of cycles between advection (default = 0).

METH
Advection method.

0
Donor cell + Half Index Shift (first order accurate) (default).

1
Van Leer + Half Index Shift (second order accurate).

Command Default

The Lagrangian formulation is used for all elements by default.

Notes

This command sets global ALE controls in an explicit dynamic analysis. These ALE controls apply to all PLANE162 or SOLID164 elements in the model that are flagged to use the ALE formulation (KEYPOPT(5) = 1). To activate the ALE formulation, you must specify the number of cycles between advection on this command and at least one smoothing weight factor on the **EDALE** command. See Arbitrary Lagrangian-Eulerian Formulation in the *ANSYS LS-DYNA User's Guide* for more information.

To see the current **EDGCALE** settings, issue the command **EDALE**,LIST.

The **EDGCALE** command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Analysis Options>ALE Options>Define

/EDGE, *WN*, *KEY*, *ANGLE*

Displays only the "edges" of an object.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

KEY
Edge key:

Elements Plots

0
Display common lines between all adjacent element faces.

1
Display only the common lines between non-coplanar faces (that is, show only the edges).

Contour Plots

0
Display only the common lines between non-coplanar faces.

1
Display common lines between all element faces.

ANGLE
Largest angle between two faces for which the faces are considered to be coplanar (0° to 180°). Defaults to 45°. A smaller angle produces more edges, a larger angle produces fewer edges.

Command Default

For element plots, display common lines between all adjacent element faces. For contour plots, display only the common lines between non-coplanar faces.

Notes

The *ANGLE* field is used in PowerGraphics to determine geometric discontinuities. It is a tolerance measure for the differences between the normals of the surfaces being considered. Values within the tolerance are accepted as coplanar (geometrically continuous).

A surface can be displayed as an edge outline without interior detail. This is useful for both geometry and post-processing displays. Element outlines are normally shown as solid lines for geometry and displacement displays.

Lines common to adjacent "coplanar" element faces are removed from the display. Midside nodes of elements are ignored. The **/SHRINK** option is ignored with the edge option. **/EDGE** is not supported for **PLESOL** and **/ESHAPE** displays when in PowerGraphics mode [**/GRAPHICS**,POWER].

The **/EDGE** command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Edge Options

EDHGLS, *HGCO*

Specifies the hourglass coefficient for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

HGCO

Hourglass coefficient value (defaults to 0.1). Values greater than 0.15 may cause instabilities.

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Hourglass Ctrls>Global
Main Menu>Preprocessor>Material Props>Hourglass Ctrls>Global
Main Menu>Solution>Analysis Options>Hourglass Ctrls>Global
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Hourglass Ctrls>Global

---

EDHIST, *Comp*

Specifies time-history output for an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Comp

Name of the component containing nodes or elements for which output is desired. *Comp* is required.

Command Default

No time-history output is written.

Notes

The time-history output is written to the file **Jobname.HIS**. Output is written only for the nodes or elements contained in *Comp*. The data is written at time intervals specified on the **EDHTIME** command. If no time interval is specified, output is written at 1000 steps over the analysis. (See also the **EDOUT** command which controls time-history output in ascii form for an explicit dynamics analysis.)

Use **EDHIST**,LIST to list the time-history output specification. (The listing will include output requested with the **EDOUT** command.) Use **EDHIST**,DELE to delete the time-history output specification.

Jobname.HIS is a binary file that is read by the ANSYS time-history postprocessor (POST26). If LS-DYNA output has been requested on the **EDWRITE** command [**EDWRITE**,LSDYNA or **EDWRITE**,BOTH], the file **D3THDT** will also be written. **D3THDT** is a binary file that is read by the LS-POST postprocessor.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Output Controls>Select Component

EDHTIME, *NSTEP*, *DT*

Specifies the time-history output interval for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

NSTEP
Number of steps at which output is written to the time-history file, **Jobname.HIS**, and the ASCII output files. Defaults to 1000. The time increment between output is *TIME* / *NSTEP*, where *TIME* is the analysis end-time specified on the **TIME** command.

DT
Time interval at which output is written to the time-history file, **Jobname.HIS**, and the ASCII output files. If *NSTEP* is input, *DT* is ignored.

Command Default

Time-history output is written at 1000 steps over the analysis.

Notes

EDHTIME controls the number of steps at which output will be written to the time-history file, **Jobname.HIS** (see the **EDHIST** command), and any ASCII files requested on the **EDOUT** command. You can use *NSTEP* or *DT* to specify the output interval. You should not specify both quantities; if both are input, *NSTEP* will be used.

In an explicit dynamic small restart (**EDSTART**,2) or full restart analysis (**EDSTART**,3), the **EDHTIME** setting will default to the *NSTEP* or *DT* value used in the original analysis. You can issue **EDHTIME** in the restart to change this setting.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Output Controls>File Output Freq>Number of Steps
Main Menu>Solution>Output Controls>File Output Freq>Time Step Size

---

EDINT, *SHELLIP*, *BEAMIP*

Specifies number of integration points for explicit shell and beam output.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

SHELLIP

Number of shell integration points used for output (defaults to 3). For element SHELL163, each integration point is associated with a layer. *SHELLIP* must be $\geq$ 3. If *SHELLIP* = 3, results are written for the shell top, middle, and bottom. If *SHELLIP* >3, then the results for the first *SHELLIP* layers are written.

BEAMIP

Number of beam integration points used for stress output for BEAM161 (defaults to 4).

Command Default

For SHELL163, output is available for the top, middle and bottom layers. For BEAM161, stress is available at 4 integration points (top-front, top-back, bottom-front, bottom-back). For the resultant beam formulation (KEYOPT(1) = 2), there is no stress output.

Notes

The number of integration points is defined by the element real constant NIP for both the beam elements (in the cross section) and the shell elements (through the thickness).

For shell elements that have only 1 or 2 integration points (NIP = 1 or 2), use the default of *SHELLIP* = 3. If NIP = 1, the same results are reported at the top, middle, and bottom layers. If the NIP = 2, the results at the bottom correspond to integration point 1, the results at the top correspond to integration point 2, and the results at the middle are an average of the top and bottom results.

For shell elements with 2 x 2 integration points in the plane, the data from the four points are averaged, and there is a single output value for each layer.

If you set *BEAMIP* = 0, no stress output is written for BEAM161 elements. In this case, the beams will not appear in any POST1 plots because the program assumes they are failed elements.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Output Controls>Integ Pt Storage

EDIPART, *PART*, *Option*, *Cvect*, *TM*, *IRCS*, *Ivect*, *Vvect*, *CID*

Defines inertia for rigid parts in an explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

PART

Part number for which the inertia is defined (the part number must have been previously generated using the **EDPART** command). The part should be composed of a rigid material (**EDMP**,RIGID). For *Option* = ADD, you must input a value; there is no default. For *Option* = DELE or LIST, *PART* defaults to all parts.

Option

ADD

Define inertia for the specified *PART* (default).

DELE

Delete the inertia properties for the specified *PART*. The remaining fields are ignored. If *PART* is blank, inertia properties previously specified using **EDIPART** are deleted for all rigid parts.

LIST

List the inertia properties for the specified *PART*. The remaining fields are ignored. If *PART* is blank, inertia properties are listed for all rigid parts.

Cvect

The vector containing the global Cartesian coordinates of the center of mass for the part. This vector must have been previously defined with a dimension of three (***DIM** command) and filled in as shown below. If *Cvect* is blank, the global Cartesian origin (0,0,0) is used as the center of mass.

Cvect(1) -- X-coordinate of the center of mass
Cvect(2) -- Y-coordinate of the center of mass
Cvect(3) -- Z-coordinate of the center of mass

TM

Translation mass (no default, must be defined).

IRCS

Flag for inertia tensor reference coordinate system.

0 (or blank)

Global inertia tensor (default). You must supply all six inertia tensor components (see *Ivect*).

1

Principal moments of inertia with orientation vectors. You must supply IXX, IYY, IZZ (see *Ivect*) and *CID*.

Ivect

The name of a vector containing the components of the inertia tensor. This vector must have been previously defined (***DIM** command) with a dimension of six and filled in as shown below. Vector entries 2, 3, and 5 are ignored if *IRCS* = 1. There is no default for this vector; it must be specified.

Ivect(1) -- IXX component of the inertia tensor
Ivect(2) -- IXY (set this entry to zero if *IRCS* = 1)
Ivect(3) -- IXZ (set this entry to zero if *IRCS* = 1)
Ivect(4) -- IYY component of the inertia tensor
Ivect(5) -- IYZ (set this entry to zero if *IRCS* = 1)
Ivect(6) -- IZZ component of the inertia tensor

Vvect
The name of a vector containing the initial velocity (relative to the global Cartesian coordinate system) of the rigid part. This vector must have been previously defined (***DIM** command) with a dimension of six and filled in as shown below. If *Vvect* is blank, the initial velocity defaults to zero.

Vvect(1) -- Initial translational velocity of rigid body in X-direction
Vvect(2) -- Initial translational velocity of rigid body in Y-direction
Vvect(3) -- Initial translational velocity of rigid body in Z-direction
Vvect(4) -- Initial rotational velocity of rigid body about the X-axis
Vvect(5) -- Initial rotational velocity of rigid body about the Y-axis
Vvect(6) -- Initial rotational velocity of rigid body about the Z-axis

CID
Local coordinate system ID. This coordinate system must have been previously defined with the **EDLCS** command. You must input *CID* if *IRCS* = 1 (no default).

Command Default

Inertia properties are calculated by the program for all rigid parts.

Notes

The **EDIPART** command applies only to rigid parts (**EDMP**,RIGID). It allows you to input the inertia properties for the rigid part rather than having the program calculate the properties from the finite element mesh.

This command is also valid in Solution.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Inertia Options>Define Inertia
Main Menu>Preprocessor>LS-DYNA Options>Inertia Options>Delete Inertia
Main Menu>Preprocessor>LS-DYNA Options>Inertia Options>List Inertia

---

EDIS, *Option*, *PIDN*, *PIDO*

Specifies stress initialization in an explicit dynamic full restart analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed.

ADD
Define stress initialization between parts (default).

DELE
Delete stress initialization between parts.

LIST
List stress initialization between parts.

PIDN
New part ID or part assembly ID in the full restart analysis (defaults to all parts in the model).

PIDO
Old part ID or part assembly ID in the previous analysis, (default to *PIDN*).

Command Default

No stress initialization is performed.

Notes

The **EDIS** command is only valid in an explicit dynamic full restart analysis (**EDSTART**,3). (**EDIS** is ignored if it is not preceded by the **EDSTART**,3 command.) Use **EDIS** to specify which parts and/or part assemblies should undergo stress initialization in the restart based on the stresses from the previous analysis. You can specify stress initialization for multiple parts (or part assemblies) by issuing **EDIS** multiple times. If you issue **EDIS** with no arguments, stress initialization is performed for all parts in the restart analysis that have a corresponding part (having the same part ID) in the previous analysis.

In a full restart analysis, the complete database is written as an LS-DYNA input file, **Jobname_*nn*.K**. When the LS-DYNA solution begins, LS-DYNA performs the stress initialization using file **Jobname_*nn*.K** and the restart dump file (**d3dump*nn*** specified on the **EDSTART** command) from the previous analysis. At the end of initialization, all the parts that were specified by the **EDIS** commands are initialized from the data saved in the restart dump file. In order for the stress initialization to be performed successfully, the new parts in the full restart analysis and the old parts in the previous analysis must have the same number of elements, same element order, and same element topology. (The parts may have different identifying numbers.) If this is not the case, the stresses cannot be initialized. If part assemblies are used, the part assemblies must contain the same number of parts. (See A Full Restart in the *ANSYS LS-DYNA User's Guide* for more details).

Menu Paths

Main Menu>Solution>Analysis Options>Initial Stress

EDLCS, *Option*, *CID*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*, *X3*, *Y3*, *Z3*

Defines a local coordinate system for use in explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed:

ADD
Define a coordinate system (default).

DELE
Delete a coordinate system. If *CID* is blank, all coordinate systems are deleted.

LIST
List defined coordinate systems. If *CID* is blank, all coordinate systems are listed.

CID
Coordinate system ID.

X1, *Y1*, *Z1*
X, Y, and Z coordinates of a point on the local x-axis.

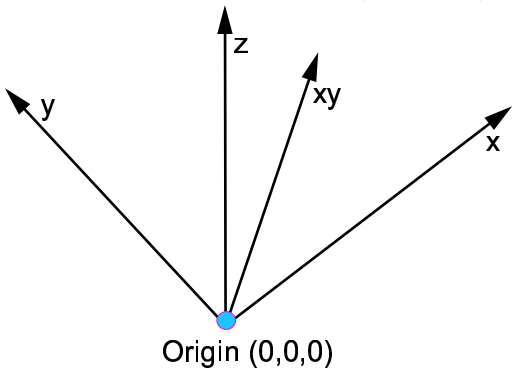
X2, *Y2*, *Z2*
X, Y, and Z coordinates of a point on the local x-y plane.

X3, *Y3*, *Z3*
X, Y, and Z coordinates of the origin. *X3*, *Y3*, and *Z3* all default to zero.

Notes

Local coordinate systems defined by this command are used in an explicit dynamic analysis. For example, a local coordinate system may be used when defining orthotropic material properties (see **EDMP**).

The coordinate system is defined by 2 vectors, one from the origin (*X3*, *Y3*, *Z3*) to a point on the x-axis (*X1*, *Y1*, *Z1*), and one from the origin to a point on the x-y plane (*X2*, *Y2*, *Z2*). The cross product of these two vectors determines the z-axis, and the cross product of the z-axis vector and x-axis vector determines the y-axis. If *X3*, *Y3*, and *Z3* are not specified, the global origin (0,0,0) is used by default (as shown in the figure below).



The x-axis vector and the xy vector should be separated by a reasonable angle to avoid numerical inaccuracies.

When you use the local coordinate system (defined by the **EDLCS** command) to define a load (**EDLOAD** command), the direction of the load will depend on the load type. For force and moment loads (*Lab* = FX, MX, etc. on **EDLOAD**), the load will be applied in the direction of the local coordinate system defined by **EDLCS**. For prescribed motion degrees of freedom (*Lab* = UX, ROTX, VX, AX, etc. on **EDLOAD**), the motion will act in the direction of a vector from point (*X1*, *Y1*, *Z1*) to point (*Z2*, *Y2*, *Z2*) as input on **EDLCS**. See the **EDLOAD** command for more information.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Local CS>Create Local CS
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Local CS>Delete
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Local CS>List
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>Local CS>Create Local CS
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>Local CS>Delete
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>Local CS>List
Main Menu>Preprocessor>Material Props>Local CS>Create Local CS
Main Menu>Preprocessor>Material Props>Local CS>Delete

Main Menu>Preprocessor>Material Props>Local CS>List
Main Menu>Solution>Constraints>Apply>Local CS>Create Local CS
Main Menu>Solution>Constraints>Apply>Local CS>Delete
Main Menu>Solution>Constraints>Apply>Local CS>List
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Local CS>Create Local CS
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Local CS>Delete
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Local CS>List

EDLOAD, *Option*, *Lab*, *KEY*, *Cname*, *Par1*, *Par2*, *PHASE*, *LCID*, *SCALE*, *BTIME*, *DTIME*

Specifies loads for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the load option to be performed.

ADD
Define a load (default). If *Option* = ADD, *Cname* must be a valid node or element component name (or PART number). You must also specify a load curve using *Par1* and *Par2* (previously defined array parameters) or *LCID* (a previously defined load curve).

DELE
Delete specified load. If *Lab* and *Cname* are blank, all loads are deleted. *Par1*, *Par2*, *PHASE*, and *LCID* are ignored for this option.

LIST
List specified load. If *Lab* and *Cname* are blank, all loads are listed. *Par1*, *Par2*, *PHASE*, and *LCID* are ignored for this option.

Lab
Valid load labels for loads applied to nodes:

FX, FY, FZ
Forces.

MX, MY, MZ
Moments.

UX, UY, UZ
Displacements.

ROTX, ROTY, ROTZ
Rotations.

VX, VY, VZ
Velocities.

OMGX, OMGY, OMGZ
Angular velocities.

AX, AY, AZ
Accelerations (on nodes).

ACLX, ACLY, ACLZ
Base accelerations.

TEMP
Temperature.

Valid load labels for loads applied to elements:

PRESS
Pressure (applied to an element).

Valid load labels for loads applied to rigid bodies:

RBFX, RBFY, RBFZ
Forces on rigid bodies.

RBMX, RBMY, RBMZ
Moments on rigid bodies.

RBUX, RBUY, RBUZ
Displacements on rigid bodies.

RBRX, RBRY, RBRZ
Rotations on rigid bodies.

RBVX, RBVY, RBVZ
Velocities on rigid bodies.

RBOX, RBOY, RBOZ
Angular velocities on rigid bodies.

KEY
When *Lab* = PRESS, *KEY* = Load key (face number) associated with a surface pressure load. Load keys (1,2,3, etc.) are listed under "Surface Loads" in the input data tables for each element type in the *ANSYS Elements Reference*.

For most other values of *Lab*, *KEY* is a coordinate system identification number, CID from the **EDLCS** command. The CID will represent either a local coordinate system (used for loads labels FX, MX, etc.) or a direction vector (used for prescribed motion labels UX, ROTX, VX, AX, etc.). See the Notes section for more information on how the CID is used. If the load is in the global coordinate system, set *KEY* equal to zero, or leave it blank. Some load types do not support the CID key; see Birth Time, Death Time, and CID Support in the Notes section for more information.

For *Lab* = PRESS, *KEY* defaults to 1 when *KEY* is blank or when *KEY* = zero. For all other values of *Lab*, *KEY* defaults to zero.

Cname
Name of existing component [**CM**] or PART number [**EDPART**] to which this load is to be applied. For all load labels except the pressure load (*Lab* = PRESS) and the rigid body loads (*Lab* = RB*xx*), the component must consist of nodes. For pressure loads, the component must consist of elements. For rigid body loads, a part number must be input instead of a component name. The part number must correspond to a set of elements that has been identified as a rigid body [**EDMP**,RIGID,*MAT*].

Par1
Name of user-defined array parameter that contains the time values of the load.

Par2
Name of user-defined array parameter that contains the "data" values of the load corresponding to the time values in *Par1*.

Note — If the length of *Par1* and *Par2* are different, the shortest length will be used.

PHASE
Phase of the analysis in which the load curve is to be used.

0
Curve is used in transient analysis only (default).

1
Curve is used in stress initialization or dynamic relaxation only.

2
Curve is used in both stress initialization (or dynamic relaxation) and transient analysis.

LCID
Data curve ID number representing the load curve to be applied. The load curve must have been previously defined using the **EDCURVE** command. If *LCID* is specified, *Par1* and *Par2* must be left blank (in the GUI, select "None" for *Par1* and *Par2*).

SCALE
Load curve scale factor applied to the specified load curve. The scale value is applied to the data in *Par2* or to the ordinate data in the load curve specified by *LCID*.

BTIME
Birth time, or time when imposed motion is activated. The default is 0.0. Some load types do not support birth and death time; see Birth Time, Death Time, and CID Support in the Notes section for more information.

DTIME
Death time, or time when imposed motion is removed. The default is 1×10^{38}. Some load types do not support birth and death time; see Birth Time, Death Time, and CID Support in the Notes section for more information.

Notes

If a component name is input (*Cname*) and the specified component definition is changed before the **SOLVE** command, the last definition will be used.

You can specify the load data by inputting *LCID* (the ID number of a previously defined load curve) or by inputting the two array parameters *Par1* and *Par2* (which contain time and load values, respectively). The input for *Par1* and *Par2* may be a single column array parameter, or a specific column from a multi-column array parameter. A starting array element number can be specified for *Par1* and *Par2*; if none is specified, array element 1 is used by default.

Note — You cannot update a previously defined load by changing the array parameters that were input as *Par1* and *Par2*. The load definition is written to the database at the time **EDLOAD** is issued. Therefore, subsequent changes to the array parameters that were used as input on **EDLOAD** will not affect the load curve definition. If you need to change the load definition, you must delete the load (**EDLOAD**,DELE) and define it again.

EDLOAD automatically assigns a load number to each defined load. Use **EDLOAD**,LIST to obtain a list of loads and their corresponding load numbers. You can plot a load curve by inputting the load number on the **EDPL** command. The load numbers may change when loads are deleted (**EDLOAD**,DELE). Therefore, you should obtain a current list of load numbers (**EDLOAD**,LIST) before plotting a load curve.

For prescribed motion, we recommend that you specify velocity time histories instead of displacement time histories. Also, you should not specify nonzero initial displacements. A piecewise linear displacement time history may lead to discontinuous velocities and infinite accelerations.

By default, the load will be applied in the global Cartesian direction. You can define the load in a different direction by inputting a CID (coordinate system ID) value in the *KEY* field. The CID must be previously defined using the **EDLCS** command. For load labels (*Lab* = FX, FY, FZ, MX, MY, MZ, RBFX, RBFY, RBFZ, RBMX, RBMY, RBMZ), the load will be applied in the direction of the local coordinate system defined by **EDLCS**. For prescribed motion degrees of freedom labels (Lab = UX, UY, UZ, ROTX, ROTY, ROTZ, VX, VY, VZ, AX, AY, AZ, RBUX, RBUY, RBUZ, RBRX, RBRY, RBRZ, RBVX, RBVY, RBVZ, RBOX, RBOY, RBOZ), the motion will act in the direction of a vector defined by two points input on the **EDLCS** command. The origin and terminus ends of the vector are defined by the *X1*, *Y1*, *Z1* and *X2*, *Y2*, *Z2* fields, respectively, of **EDLCS**.

For *Lab* = OMGX, OMGY, and OMGZ, you may need to specify the origin location of the acceleration coordinate system [**CGLOC**].

When applying a temperature load (*Lab* = TEMP), you may also need to define a reference temperature via the **TREF** command. The thermal loading is defined as the difference between the applied temperature and the reference temperature. Note that **EDLOAD**,LIST will list only the temperature values specified on **EDLOAD**, not the temperature difference.

When applying loads to axisymmetric PLANE162 elements, the load may be interpreted differently depending on whether you use the area weighted or volume weighted option (KEYOPT(2)). See the PLANE162 element description in the *ANSYS Elements Reference* for details.

Use *PHASE* = 0 when you are using the LS-DYNA solver to conduct a transient explicit analysis only or when you are conducting a sequential implicit/explicit analysis, in which the ANSYS (implicit) resulting displacements (stored in the `drelax' file from the **REXPORT** command) are used to preload the explicit model [**EDDRELAX**,ANSYS]

Use *PHASE* = 1 or 2 when you need to use LS-DYNA to preload the model (as opposed to ANSYS) before running the transient portion of the analysis. *PHASE* = 1 applies the load initially and then immediately removes the load. As a result, the load is removed, and the structure vibrates freely. *PHASE* = 2 applies the load and then continues to apply the load over the course of the transient analysis, so that the transient analysis includes the effect of the initial loading and continues to account for the initial loading.

Birth and Death times, as well as the CID key are supported only for the **EDLOAD** labels specified with a Yes in the following table.

Birth Time, Death Time, and CID Support

EDLOAD Label	Birth Time	Death Time	*KEY* = CID
FX, FY, FZ	No	No	Yes
MX, MY, MZ	No	No	Yes
UX, UY, UZ	Yes	Yes	Yes
ROTX, ROTY, ROTZ	Yes	Yes	Yes
VX, VY, VZ	Yes	Yes	Yes
OMGX, OMGY, OMGZ	No	No	No
AX, AY, AZ	Yes	Yes	Yes
ACLX, ACLY, ACLZ	No	No	No
TEMP	No	No	No

EDLOAD Label	Birth Time	Death Time	*KEY* = CID
PRESS	Yes	No	No
RBFX, RBFY, RBFZ	No	No	Yes
RBMX, RBMY, RBMZ	No	No	Yes
RBUX, RBUY, RBUZ	Yes	Yes	Yes
RBRX, RBRY, RBRZ	Yes	Yes	Yes
RBVX, RBVY, RBVZ	Yes	Yes	Yes
RBOX, RBOY, RBOZ	Yes	Yes	Yes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Delete Loads>Delete All
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Delete Loads>Delete Individ
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Specify Loads
Main Menu>Solution>Loading Options>Delete Loads>Delete All
Main Menu>Solution>Loading Options>Delete Loads>Delete Individ
Main Menu>Solution>Loading Options>Specify Loads

EDMP, *Lab*, *MAT*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*

Defines material properties for an explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Lab

Valid material property label. Applicable labels are listed under "Material Properties" in the input table for each explicit dynamics element type in the *ANSYS Elements Reference*.

HGLS

Hourglass and bulk viscosity properties (valid for PLANE162, SHELL163, SOLID164 using reduced integration, and SOLID168). *VAL1* through *VAL6* are also used. For those elements using full integration, HGLS is not applicable and the input has no effect.

RIGID

Rigid body constraint (valid for LINK160, BEAM161, PLANE162, SHELL163, SOLID164, and SOLID168). *VAL1* and *VAL2* are also used.

CABLE

Cable properties (valid for LINK167). *VAL1* is optional input (see Notes).

ORTHO

Defines a material coordinate system for the orthotropic material model (valid for PLANE162, SHELL163, SOLID164, and SOLID168) or the anisotropic material model (valid for SOLID164 and SOLID168). *VAL1* is also used.

FLUID

Fluid properties (valid for PLANE162, SOLID164, and SOLID168). *VAL1* is optional input (see Notes).

MAT
Material reference number (defaults to the current *MAT* setting on **MAT** command).

VAL1, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*
Additional input for specified *Lab* material property. The meaning of *VAL1* through *VAL6* will vary, depending on *Lab*. See the table below for *VAL1* through *VAL6* definitions.

VAL1-VAL6 Definitions

Additional input for hourglass and bulk viscosity properties (*Lab* = HGLS).

VAL1
Hourglass control type. For solid elements (PLANE162, SOLID164, and SOLID168), 5 options are available. For quadrilateral shell and membrane elements (SHELL163) with reduced integration, the hourglass control is based on the formulation of Belytschko and Tsay; i.e., options 1-3 are identical and options 4-5 are identical.

0, 1
Standard LS-DYNA viscous form (default).

2
Flanagan-Belytschko viscous form.

3
Flanagan-Belytschko viscous form with exact volume integration for solid elements.

4
Flanagan-Belytschko stiffness form.

5
Flanagan-Belytschko stiffness form with exact volume integration for solid elements.

VAL2
Hourglass coefficient. (Defaults to 0.1.) Values greater than 0.15 may cause instabilities. The recommended default applies to all options. The stiffness forms can stiffen the response (especially if deformations are large) and, therefore, should be used with care. For the shell and membrane elements, the value input for *VAL1* is the membrane hourglass coefficient. *VAL5* and *VAL6* can also be input, but generally *VAL2* = *VAL5* = *VAL6* is adequate.

VAL3
Quadratic bulk viscosity coefficient. (Defaults to 1.5.)

VAL4
Linear bulk viscosity coefficient. (Defaults to 0.06.)

VAL5
Hourglass coefficient for shell bending. (Defaults to *VAL2*.)

VAL6
Hourglass coefficient for shell warping. (Defaults to *VAL2*.)

Additional input for rigid body constraint (*Lab* = RIGID).

VAL1
Translational constraint parameter (relative to global Cartesian coordinates).

0
No constraints (default).

1
Constrained X displacement.

2
Constrained Y displacement.

3
Constrained Z displacement.

4
Constrained X and Y displacements.

5
Constrained Y and Z displacements.

6
Constrained Z and X displacements.

7
Constrained X, Y, and Z displacements.

VAL2
Rotational constraint parameter (relative to global Cartesian coordinates).

0
No constraints (default).

1
Constrained X rotation.

2
Constrained Y rotation.

3
Constrained Z rotation.

4
Constrained X and Y rotations.

5
Constrained Y and Z rotations.

6
Constrained Z and X rotations.

7
Constrained X, Y, and Z rotations.

Additional input for cable properties (*Lab* = CABLE).

VAL1
Load curve ID defining engineering stress versus engineering strain (i.e., change in length over the initial length). If *VAL1* and Young's modulus [**MP**,EX] are input, the load curve corresponding to *VAL1* will be used and Young's modulus will be ignored.

Additional input for material coordinate system (*Lab* = ORTHO).

VAL1
Coordinate system ID number from the **EDLCS** command. This coordinate system will be used to orient the orthotropic or anisotropic materials associated with the material number, *MAT*.

Additional input for fluid material properties (*Lab* = FLUID).

VAL1
Bulk modulus of fluid. If *VAL1* is not input, the bulk modulus will be calculated from the elastic modulus (EX) and Poisson's ratio (NUXY).

Notes

For *Lab* = RIGID, you must specify elastic modulus (EX), density (DENS), and Poisson's ratio (NUXY) [**MP** command]. For *Lab* = CABLE, you must specify density (DENS) and one of the following: Young's modulus (EX) or an engineering stress-strain curve (*VAL1*). For *Lab* = FLUID, you must specify either the bulk modulus (*VAL1*) or both Young's modulus (EX) and Poisson's ratio (NUXY) (if all three are specified, only *VAL1* will be used).

After you define a rigid body using **EDMP**,RIGID, you may assign inertia properties to that rigid body using the **EDIPART** command.

EDMP,ORTHO is required for orthotropic or anisotropic material properties that are not aligned with the global Cartesian coordinate system.

Via the GUI, *Lab* = RIGID, CABLE, ORTHO, and FLUID are available through the material model interface. See Defining Explicit Dynamics Material Models in the *ANSYS LS-DYNA User's Guide* for more information.

Use the **MPLIST** and **MPDELE** commands to list and delete materials defined by the **EDMP** command.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Hourglass Ctrls>Local
Main Menu>Preprocessor>Material Props>Hourglass Ctrls>Local
Main Menu>Solution>Analysis Options>Hourglass Ctrls>Local
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Hourglass Ctrls>Local

---

EDNB, *Option*, *Cname*, *AD*, *AS*

Defines a nonreflecting boundary in an explicit dynamic analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the nonreflecting boundary option to be performed.

ADD
Define a nonreflecting boundary (default).

DELE
Delete a nonreflecting boundary.

LIST
List all defined nonreflecting boundaries (remaining fields are ignored).

Cname
Name of existing nodal component to which the nonreflecting boundary is to be added or deleted. For *Option* = DELE, use *Cname* = ALL to delete all defined nonreflecting boundaries.

AD
Activation flag for dilatational waves (dampers normal to waves).

0
Dilatational activation flag is off (default).

1
Dilatational activation flag is on.

AS
Activation flag for shear waves (dampers tangent to waves).

0
Shear activation flag is off (default).

1
Shear activation flag is on.

Notes

Nonreflecting boundaries can be defined on the external surfaces of SOLID164 and SOLID168 elements that are being used to model an infinite domain. They are typically used in geomechanical applications to limit the size of the model. For example, when a half space is being modeled with a finite geometry, the nonreflecting boundary option can be used to prevent artificial stress wave reflections generated at the boundary from reentering the model and contaminating the results.

When using nonreflecting boundaries, you should not constrain the nodes at the boundary; doing so would negate the presence of the dampers. Usually, the large mass of the finite domain is sufficient to resist motion.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>Non-Refl Bndry
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Non-Refl Bndry>Delete All
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Non-Refl Bndry>Delete Individ
Main Menu>Solution>Constraints>Apply>Non-Refl Bndry
Main Menu>Solution>Constraints>Delete>Non-Refl Bndry>Delete All
Main Menu>Solution>Constraints>Delete>Non-Refl Bndry>Delete Individ

EDNDTSD, *Vect1*, *Vect2*, *DATAP*, *FITPT*, *Vect3*, *Vect4*, *DISP*

Allows smoothing of noisy data for explicit dynamics analyses and provides a graphical representation of the data.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Vect1
Name of the first vector that contains the noisy data set (i.e., independent variable). You must create and fill this vector before issuing **EDNDTSD**.

Vect2
Name of the second vector that contains the dependent set of data. Must be the same length as the first vector. You must create and fill this vector before issuing **EDNDTSD**.

DATAP
Number of data points to be fitted, starting from the beginning of the vector. If left blank, the entire vector will be fitted. The maximum number of data points is 100,000 (or greater, depending on the memory of the computer).

FITPT
Order of the fitting curve that will be used as a smooth representation of the data. This number should be less than or equal to the number of the data points. Default (blank) is one-half the number of data points. Maximum number of smoothed data fitting order is the number of data points up to 50. Depending on this number, the smoothed curve will be one of the following:

1
Curve is the absolute average of all of the data points.

2
Curve is the least square average of all of the data points.

3 or more
Curve is a polynomial of the order (n-1), where n is the number of data fitting order points.

Vect3
Name of the vector that contains the smoothed data of the independent variable. This vector should have a length equal to or greater than the number of smoothed data points. In batch (command) mode, you must create this vector before issuing the **EDNDTSD** command. In interactive mode, the GUI automatically creates this vector (if it does not exist). If you do not specify a vector name, the GUI will name the vector smth_ind.

Vect4
Name of the vector that contains the smoothed data of the dependent variable. This vector must be the same length as *Vect3*. In batch (command) mode, you must create this vector before issuing the **EDNDTSD** command. In interactive mode, the GUI automatically creates this vector (if it does not exist). If you do not specify a vector name, the GUI will name the vector smth_dep.

DISP
Specifies how you want to display data. No default; you must specify an option.

1
Unsmoothed data only

2
Smoothed data only

3
Both smoothed and unsmoothed data

Notes

You can control the attributes of the graph using standard ANSYS controls (**/GRID**, **/GTHK**, **/COLOR**, etc.). If working interactively, these controls appear in this dialog box for convenience, as well as in their standard dialog boxes. You must always create *Vect1* and *Vect2* (using ***DIM**) and fill these vectors before smoothing the data. If you're working interactively, ANSYS automatically creates *Vect3* and *Vect4*, but if you're working in batch (command) mode, you must create *Vect3* and *Vect4* (using ***DIM**) before issuing **EDNDTSD**. *Vect3* and *Vect4* are then filled automatically by ANSYS. In addition, ANSYS creates an additional TABLE type array that contains the smoothed array and the unsmoothed data to allow for plotting later with ***VPLOT**. Column 1 in this table corresponds to *Vect1*, column 2 to *Vect2*, and column 3 to *Vect4*. This array is named *Vect3*_SMOOTH, up to a limit of 32 characters. For example, if the array name is X1, the table name is X1_SMOOTH.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Smooth Data
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Smooth Data
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Smooth Data
Main Menu>Solution>Loading Options>Smooth Data
Main Menu>TimeHist Postpro>Smooth Data

EDNROT, *Option*, *CID*, *Cname*, *DOF1*, *DOF2*, *DOF3*, *DOF4*, *DOF5*, *DOF6*

Applies a rotated coordinate nodal constraint in an explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed:

ADD
Add a rotated nodal coordinate constraint (default).

DELE
Delete specified rotated nodal coordinate constraints.

LIST
List all rotated nodal coordinate constraints.

CID
Coordinate system ID for which rotated nodal constraints will be added or deleted. The *CID* must have been previously defined with the **EDLCS** command. If *Option* = DELE, use *CID* = ALL to delete all previously specified nodal constraints.

Cname
Nodal component set to which the rotated coordinate constraint will be applied. *Cname* must be previously specified using the **CM** command.

D0F1, *D0F2*, *D0F3*, *D0F4*, *D0F5*, *D0F6*
Degrees of freedom for which the rotated nodal constraint will be applied. Valid degree of freedom labels include UX, UY, UZ, ROTX, ROTY, and ROTZ. If *D0F1* = ALL, rotated nodal constraints will be applied to all degrees of freedom.

Notes

Constraints applied with **EDNROT** are zero displacement constraints.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Constraints>Apply>Rotated Nodal
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Rotated Nodal>Delete All
Main Menu>Preprocessor>LS-DYNA Options>Constraints>Delete>Rotated Nodal>Delete Individ
Main Menu>Solution>Constraints>Apply>Rotated Nodal
Main Menu>Solution>Constraints>Delete>Rotated Nodal>Delete All
Main Menu>Solution>Constraints>Delete>Rotated Nodal>Delete Individ

---

EDOPT, *Option*, --, *Value*

Specifies the type of output for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed:

ADD
Define an output type specification (default).

DELE
Delete an output type specification.

LIST
List the current output type specification.

--
This field is reserved for future use.

Value
Label identifying the type of output that the LS-DYNA solver should produce:

ANSYS
Write results files for the ANSYS postprocessors (default). The files that will be written are **Jobname.RST** and **Jobname.HIS** (see Notes below).

LSDYNA
Write results files for the LS-DYNA postprocessor (LS-POST). The files that will be written are **D3PLOT**, and files specified by **EDOUT** and **EDHIST** (see Notes below).

BOTH
Write results files for both ANSYS and LS-DYNA postprocessors.

Command Default

Output is written for the ANSYS postprocessors only.

Notes

By default, LS-DYNA will write the ANSYS results file **Jobname.RST** (see the **EDRST** command.) If **Jobname.HIS** is desired, you must also issue **EDHIST**.

Value = LSDYNA or BOTH will cause LS-DYNA to write results files for the LS-POST postprocessor. The **D3PLOT** file is always written for these two options. If other LS-POST files are desired, you must issue the appropriate **EDHIST** and **EDOUT** commands.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Output Controls>Output File Types

EDOUT, *Option*

Specifies time-history output (ASCII format) for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option

Output data option. Each option corresponds to a separate file that is written by the LS-DYNA solver. If *Option* = ALL, all files except NODOUT and ELOUT are written. Valid options are:

GLSTAT
: Global data (default).

BNDOUT
: Boundary condition forces and energy.

RWFORC
: Wall force.

DEFORC
: Discrete element data.

MATSUM
: Material energies data.

NCFORC
: Nodal interface forces.

RCFORC
: Resultant interface force data.

DEFGEO
: Deformed geometry data.

SPCFORC
: SPC reaction force data.

SWFORC
Nodal constraint reaction force data (spotwelds and rivets).

RBDOUT
Rigid body data.

GCEOUT
Geometry contact entities.

SLEOUT
Sliding interface energy.

JNTFORC
Joint force data.

NODOUT
Nodal data.

ELOUT
Element data.

Command Default

None of the above output is written.

Notes

This command specifies output to be written during an explicit dynamics solution. The data corresponding to each *Option* is written to a separate ASCII file having the same name as the *Option* label. The data is written for the entire model at time intervals specified by the **EDHTIME** command. If no time interval is specified, output is written at 1000 steps over the analysis. (See also the **EDHIST** command which specifies time-history output for a portion of the model.) The data written to the **MATSUM** file is actually for each PART number (**EDPART**) at time intervals specified by the **EDHTIME** command, but the data is listed following the Mat no. in the file.

For *Option* = NODOUT and ELOUT, you must specify a component; you must issue **EDHIST** before issuing **EDOUT**,NODOUT or **EDOUT**,ELOUT.

Use **EDOUT**,LIST to list the current time-history output specifications. (The listing will include output requested with the **EDHIST** command.) Use **EDOUT**,DELE to delete all output specifications that have been defined with the **EDOUT** command.

In order for the specified output files to be written, you must also request that explicit dynamics results be written to an LS-DYNA output file [**EDWRITE**,LSDYNA or **EDWRITE**,BOTH].

In an explicit dynamic small restart analysis (**EDSTART**,2) or full restart analysis (**EDSTART**,3), the same ASCII files that were requested for the original analysis are written by default for the restart. You can request different files by issuing the appropriate **EDOUT** commands in the restart analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Output Controls>ASCII Output

EDPART, *Option*, *PARTID*, *Cname*

Configures parts for an explicit dynamics analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option

Option used to organize parts. (No default; *Option* must be specified.)

CREATE

Creates new PART IDs assigned to groups of elements with unique combinations of MAT, TYPE, and REAL set numbers. If this option is issued repeatedly, the part list is overwritten, except for PART IDs created with the ADD option. Remaining fields are ignored for this option.

UPDATE

Updates the PART IDs for the element groups without changing the order of the existing part list. If elements are redefined (or new elements are created) with different MAT, TYPE, or REAL set numbers, then use this option to create an updated list of PART IDs. Remaining fields are ignored for this option.

ADD

Assigns a user-specified PART ID (*PARTID*) to the elements contained in the element component *Cname*, or to the currently selected set of elements if *Cname* = ALL. Use this option to assign a specific PART ID to an element group that has the same combination of MAT, TYPE, and REAL set numbers. An UPDATE operation is automatically performed on the currently selected set of elements immediately following the ADD operation.

DELE

Deletes a PART ID assigned by the ADD option. *PARTID* is also required. An UPDATE operation is automatically performed on the currently selected set of elements immediately following the DELE operation.

LIST

Lists the PART IDs for the element groups. The part list consists of five columns of numbers, one each for PART, MAT, TYPE, and REAL numbers, and one to indicate if the PART ID is used (including how many elements use it). The part list is based on the last CREATE or UPDATE operation. Remaining fields are ignored for this option.

PARTID

A positive integer to be used as PART ID for the elements specified by *Cname* (no default). The number input must not be currently used for an existing part (except when *Option* = DELE). Any previously defined PART IDs for the elements, whether assigned by the user or created by ANSYS LS-DYNA, will be overwritten. The user-specified PART ID will not be changed by subsequent **EDPART**,CREATE or **EDPART**,UPDATE commands.

Cname

Element component name for user-specified PART ID definition (*Option* = ADD). If *Cname* = ALL (default), all currently selected elements are considered for the part. The elements in the element component (or the currently selected set of elements if *Cname* = ALL or blank) must have the same combination of MAT, TYPE, and REAL set numbers, or the ADD option will be ignored.

Notes

Certain ANSYS LS-DYNA commands (such as **EDCGEN**, **EDLOAD**, **EDREAD**, etc.) refer to PART IDs. You must define PART IDs (**EDPART**,CREATE or **EDPART**,ADD) before using these commands.

If parts are repeatedly created using *Option* = CREATE, the part list is continuously overwritten. This may cause problems for previously defined commands that reference a part number that has changed. To avoid this problem, the part list should be updated (*Option* = UPDATE) rather than recreated to obtain the current part list.

EDPART,ADD allows you to assign a specific part number to a group of elements instead of a number generated by the ANSYS LS-DYNA program. The user-specified PART IDs will not be changed by subsequent **EDPART**,CREATE or **EDPART**,UPDATE commands. Thus, you can use **EDPART**,ADD to specify PART IDs for some element groups, and use **EDPART**,CREATE or **EDPART**,UPDATE to assign PART IDs for the remaining element groups. Use **EDPART**,DELE to delete a PART ID generated by the ADD option. In this case, ANSYS LS-DYNA will generate a new PART ID for those elements associated with the deleted PART ID.

After creating or updating the part list, use **EDPART**,LIST to list the PART IDs and choose the correct one for use with other ANSYS LS-DYNA commands. For a detailed discussion on PART IDs, see The Definition of Part in the *ANSYS LS-DYNA User's Guide*.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Parts Options

EDPC, *MIN*, *MAX*, *INC*

Selects and plots explicit dynamic contact entities.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

MIN
Minimum contact entity number to be selected and plotted (default = 1).

MAX
Maximum contact entity number to be selected and plotted (default = *MIN*).

INC
Contact entity number increment (default = 1).

Notes

EDPC invokes an ANSYS macro which selects and plots explicit dynamic contact entities. The plot will consist of nodes or elements, depending on the method (node components or parts) that was used to define the contact surfaces (see the **EDCGEN** command). For single surface contact definitions, all external surfaces within the model are plotted.

Note — **EDPC** changes the selected set of nodes and elements. After plotting contact entities, you must reselect all nodes and elements (**NSEL** and **ESEL**) required for subsequent operations, such as **SOLVE**

Use the **EDCLIST** command to list the contact entity numbers for all defined contact.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Contact>Select and Plot

---

EDPL, *LDNUM*

Plots a time dependent load curve in an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

LDNUM
Load number.

Notes

EDPL invokes an ANSYS macro which produces a load vs. time graph for a load defined with the **EDLOAD** command. Only one load curve can be plotted at a time. Use **EDLOAD**,LIST to obtain a list of loads and corresponding load numbers.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Plot Load Curve
Main Menu>Solution>Loading Options>Plot Load Curve

EDPVEL, *Option*, *PID*, *VX*, *VY*, *VZ*, *OMEGAX*, *OMEGAY*, *OMEGAZ*, *XC*, *YC*, *ZC*, *ANGX*, *ANGY*, *ANGZ*

Applies initial velocities to parts or part assemblies in an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed.

VGEN
Define initial velocities for the part or part assembly based on translational velocities (relative to global Cartesian) and the rotational velocity about an arbitrary axis. For this option, use the fields *VX*, *VY*, *VZ* to specify the translational velocities, and use *OMEGAX*, *XC*, *YC*, *ZC*, *ANGX*, *ANGY*, *ANGZ* to specify the rotational velocity and the axis of rotation.

VELO
Define initial velocity for the part or part assembly based on translational velocities and nodal rotational velocities input relative to the global Cartesian axes. For this option, use the following fields to define the initial velocity: *VX*, *VY*, *VZ*, *OMEGAX*, *OMEGAY*, *OMEGAZ*.

LIST
List initial velocity for the part or part assembly specified by *PID*. If *PID* is blank, all initial velocities defined on parts and part assemblies are listed. Remaining fields are ignored for this option.

DELE
Delete initial velocity defined for the part or part assembly specified by *PID*. If *PID* is blank, all initial velocities defined on parts and part assemblies are deleted. Remaining fields are ignored for this option.

PID
Part ID or part assembly ID to which the initial velocity is to be applied. The part or assembly ID must be defined (**EDPART** or **EDASMP**) before issuing this command.

VX
Initial velocity in X direction. Defaults to 0.

VY
Initial velocity in Y direction. Defaults to 0.

VZ
Initial velocity in Z direction. Defaults to 0.

OMEGAX
For *Option* = VGEN, *OMEGAX* is the initial rotational velocity of the part or part assembly about the specified rotational axis. For *Option* = VELO, *OMEGAX* is the initial nodal rotational velocity about the X-axis. *OMEGAX* defaults to 0.

OMEGAY
Initial nodal rotational velocity about the Y-axis (used only if *Option* = VELO). Defaults to 0.

OMEGAZ
Initial nodal rotational velocity about the Z-axis (used only if *Option* = VELO). Defaults to 0.

The remaining fields are used only if *Option* = VGEN.

XC
X coordinate on rotational axis. Defaults to 0.

YC
Y coordinate on rotational axis. Defaults to 0.

ZC
Z coordinate on rotational axis. Defaults to 0.

ANGX
Angle relative to global X-axis. Defaults to 0.

ANGY
Angle relative to global Y-axis. Defaults to 0.

ANGZ
Angle relative to global Z-axis. Defaults to 0.

Notes

You cannot mix the two methods of initial velocity input (*Option* = VELO and *Option* = VGEN) in the same analysis. You must use only one method for all initial velocity definitions.

The VGEN and VELO methods differ in how the rotational velocity is defined. Use *Option* = VGEN to input the initial velocities of a rotating part or part assembly. Use *Option* = VELO to apply the rotations directly to the nodes' rotation degrees of freedom. Since only shell and beam elements have rotation degrees of freedom, the rotations input with *Option* = VELO are only applicable to SHELL163 and BEAM161 elements. The rotational

velocities input with *Option* = VELO are ignored for nodes not having rotational degrees of freedom (such as nodes attached to a SOLID164 or SOLID168 element).

It is normally acceptable to mix nodes belonging to deformable bodies and rigid bodies in the part assembly used in an initial velocity definition. However, when defining initial velocities in an implicit-to-explicit sequential solution, this is not an acceptable practice. In order for the initial velocities to be defined correctly in this type of analysis, you must define the initial velocities on the deformable body nodes separately from the initial velocities on the rigid body nodes.

Issuing the **EDPVEL** command again for the same part or part assembly (*PID*) will overwrite previous initial velocities defined for that part or part assembly.

To set the initial velocities to zero, issue the **EDPVEL** command with only the *Option* (use VELO or VGEN) and *PID* fields specified.

In a small restart analysis (**EDSTART**,2), you can only use the *Option* = VELO method to change initial velocities. When used in a small restart, the command **EDPVEL**,VELO changes the velocity of the specified part or part assembly. If you don't change the velocity of the parts and assemblies, their velocity at the beginning of the restart will be the same as the velocity at the end of the previous analysis.

Except for the LIST option, the **EDPVEL** command is not supported in a full restart analysis (**EDSTART**,3). You can list initial velocities defined in the previous analysis with the command **EDPVEL**,LIST. However, you cannot change initial velocities for parts that existed in the previous analysis; their velocity at the beginning of the analysis will be the same as the velocity at the end of the previous analysis. In addition, you cannot define initial velocities for any parts that are added in the full restart; the velocity of new parts will be zero.

To apply initial velocities to node components or nodes, use the **EDVEL** command.

You can use **EDPVEL** and **EDVEL** in the same analysis. If a node or node component input on the **EDVEL** command shares common nodes with a part or part assembly input on the **EDPVEL** command, the initial velocities defined on the common nodes will be determined by the last command input.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Parts>Delete
Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Parts>List
Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Parts>w/Axial Rotate
Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Parts>w/Nodal Rotate
Main Menu>Solution>Initial Velocity>On Parts>Delete
Main Menu>Solution>Initial Velocity>On Parts>List
Main Menu>Solution>Initial Velocity>On Parts>w/Axial Rotate
Main Menu>Solution>Initial Velocity>On Parts>w/Nodal Rotate

EDRC, *Option*, *NRBF*, *NCSF*, --, *DTMAX*

Specifies rigid/deformable switch controls in an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying option to be performed.

ADD
Define rigid/deformable controls (default).

DELE
Delete rigid/deformable controls.

LIST
List rigid/deformable controls.

NRBF
Flag to delete/activate nodal rigid bodies. If nodal rigid bodies or generalized weld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities.

0
No change from previous status (default).

1
Delete.

2
Activate.

NCSF
Flag to delete/activate nodal constraint set. If nodal constraint/spotweld definitions are active in the deformable bodies that are switched to rigid, then the definitions should be deleted to avoid instabilities.

0
No change from previous status (default).

1
Delete.

2
Activate.

--
Unused field.

TDMAX
Maximum allowed time step after restart (no default).

Command Default

No rigid/deformable controls are defined.

Notes

This command is only valid in an explicit dynamic small restart analysis (**EDSTART**,2). Use this command when you do a rigid/deformable switch (**EDRD** command) and you want to control constraints defined by other means for the deformable body (such as nodal constraints or a weld). For example, if a deformable body has nodal constraints defined and it is switched to a rigid body, the nodal constraints should be deleted since they are invalid for the rigid body. Later on, if you want to switch the rigid body to deformable again and retain the nodal constraints, you can use **EDRC** to activate the constraints previously defined for the deformable body. Otherwise, the nodal constraints remain deactivated.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Rigid-Deformable>Controls

EDRD, *Option*, *PART*, *MRB*

Switches a part from deformable to rigid or from rigid to deformable in an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed.

D2R
Change specified part from deformable to rigid (default).

R2D
Change specified part from rigid to deformable. Use this option to switch a part back to a deformable state after it has been changed to rigid using **EDRD**,D2R.

LIST
List parts that are flagged to change from deformable to rigid or rigid to deformable.

PART
Part number for part to be changed (no default).

MRB
Part number of the master rigid body to which the part is merged. *MRB* is used only if *Option* = D2R. If *MRB* = 0 (which is the default), the part becomes an independent rigid body.

Command Default

No parts are switched.

Notes

This command is valid in a new explicit dynamic analysis or in a restart. It is only possible to switch parts (D2R or R2D) in a restart if part switching is first activated in the original analysis. If part switching is not required in the original analysis but will be used in the restart, you must issue **EDRD**,D2R with no further arguments in the original analysis. You can use the **EDRI** command to define inertia properties for newly created rigid bodies (D2R).

Parts that are defined as rigid using **EDMP**,RIGID are permanently rigid and cannot be changed to deformable.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Rigid-Deformable>Switch

EDREAD, *NSTART*, *Label*, *NUM*, *STEP1*, *STEP2*

Reads explicit dynamics output into variables for time-history postprocessing.

POST26: Set Up

<> <> <> DY <> <> <> <> <> <> ED

NSTART
Starting reference number assigned to the first variable. Allowed range is 2 (the default) to *NV* [**NUMVAR**]. (*NV* defaults to 30 for an explicit dynamics analysis.)

Label
Label identifying the output file to be read. No default.

GLSTAT
Read data from the **GLSTAT** file.

MATSUM
Read data from the **MATSUM** file.

SPCFORC
Read data from the **SPCFORC** file.

RCFORC
Read data from the **RCFORC** file.

SLEOUT
Read data from the **SLEOUT** file.

NODOUT
Read data from the **NODOUT** file.

RBDOUT
Read data from the **RBDOUT** file.

NUM
Number identifying the data set to be read in (defaults to 1). If *Label* = GLSTAT, *NUM* is ignored. If *Label* = MATSUM or RBDOUT, *NUM* is the PART number [**EDPART**] for which output is desired. If *Label* = SPCFORC or NODOUT, *NUM* is the node number for which output is desired. If *Label* = SLEOUT or RCFORC, *NUM* is the number of the contact entity for which output is desired.

STEP1, *STEP2*
Load step range of data to be read in. If *STEP1* and *STEP2* are blank, all load steps are read in.

Notes

EDREAD reads data from the specified ascii output file so that it may be used during postprocessing. After **EDREAD**, you must issue the **STORE** command to store the data in time history variables. Once stored, the variables can be viewed as plots of output item versus time.

The number of variables stored depends on the file specified. The following table shows the items in each file and the order in which they are stored. If data items were previously stored in variables *NSTART* to *NSTART*+15, they will be overwritten. If more variables are needed, change *NV* on the **NUMVAR** command. (Note that hourglass energy will not be available if it was not specified for the model [**EDENERGY**,1].)

The following items under MATSUM are listed in the **MATSUM** ASCII file (in the Mat no. field) for each part number at time intervals specified by the **EDHTIME** command. Use **EDREAD**,,MATSUM,NUM to specify the part number that corresponds to the mat number in the **MATSUM** file.

Resultant contact forces and sliding interface energies are available from the **RCFORC** and **SLEOUT** files, respectively. The **RCFORC** file is written for surface based contact types that include target and contact (master and slave) definitions. You should ensure that this file contains valid force results before issuing **EDREAD**,,RCFORC. Only the resultant contact forces on the *master* surface are available for time-history postprocessing.

Variable Number	GLSTAT	MATSUM	SPCFORC	RCFORC	SLEOUT
NSTART	Time step	Internal energy	X force	X force	Slave energy
NSTART+1	Kinetic energy	Kinetic energy	Y force	Y force	Master energy
NSTART+2	Internal energy	X-momentum	Z force	Z force	Total slave energy
NSTART+3	Spring and damper energy	Y-momentum	X moment	--	Total master energy
NSTART+4	System damping energy	Z-momentum	Y moment	--	Total energy
NSTART+5	Sliding interface energy	X-rigid-body- velocity	Z moment	--	--
NSTART+6	External work	Y-rigid-body- velocity	--	--	--
NSTART+7	Eroded kinetic energy	Z-rigid-body- velocity	--	--	--
NSTART+8	Eroded internal energy	Hourglass energy	--	--	--
NSTART+9	Total energy	--	--	--	--
NSTART+10	Total energy/initial energy	--	--	--	--
NSTART+11	Energy ratio w/o eroded energy	--	--	--	--
NSTART+12	Global X velocity	--	--	--	--
NSTART+13	Global Y velocity	--	--	--	--
NSTART+14	Global Z velocity	--	--	--	--
NSTART+15	Hourglass energy	--	--	--	--

Menu Paths

Main Menu>TimeHist Postpro>Read LSDYNA Data>GLSTAT file

Main Menu>TimeHist Postpro>Read LSDYNA Data>MATSUM file
Main Menu>TimeHist Postpro>Read LSDYNA Data>NODOUT file
Main Menu>TimeHist Postpro>Read LSDYNA Data>RBDOUT file
Main Menu>TimeHist Postpro>Read LSDYNA Data>RCFORC file
Main Menu>TimeHist Postpro>Read LSDYNA Data>SLEOUT file
Main Menu>TimeHist Postpro>Read LSDYNA Data>SPCFORC file

EDRI, *Option*, *PART*, *XC*, *YC*, *ZC*, *TM*, *IXX*, *IYY*, *IZZ*, *IXY*, *IYZ*, *IXZ*

Defines inertia properties for a new rigid body that is created when a deformable part is switched to rigid in an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed.

ADD
Define inertia for specified part (default).

DELE
Delete inertia definition for specified part.

LIST
List inertia definitions.

PART
Part number for which inertia is defined (no default).

XC, *YC*, *ZC*
X, Y, and Z-coordinates of the center of mass (no defaults).

TM
Translational mass (no default).

IXX, *IYY*, *IZZ*, *IXY*, *IYZ*, *IXZ*
Components (xx, yy, etc.) of inertia tensor. *IXX*, *IYY*, and *IZZ* must be input (no defaults). *IXY*, *IYZ*, and *IXZ* default to zero.

Command Default

Inertia properties are calculated by the program for all parts switched from deformable to rigid.

Notes

Use this command to define inertia properties for a rigid body that is created when a deformable part is switched to rigid (using the **EDRD**,D2R command) in an explicit dynamic analysis. If these properties are not defined, LS-DYNA will compute the new rigid body properties from the finite element mesh (which requires an accurate mesh representation of the body). When rigid bodies are merged to a master rigid body, the inertia properties defined for the master rigid body apply to all members of the merged set.

EDRI can only be issued in a new analysis. Therefore, if you are going to use inertia properties in a subsequent restart analysis, you must issue **EDRI** in the original analysis for the part that will later be switched to rigid in the restart.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Rigid-Deformable>Inertia Property

EDRST, *NSTEP*, *DT*

Specifies the output interval for an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

NSTEP

Number of steps at which output is written to the results file (**Jobname.RST**). Defaults to 100. When you specify *NSTEP*, *NSTEP*+2 results are written to the **Jobname.RST** file. The time interval between output is *TIME* / *NSTEP*, where *TIME* is the analysis end-time specified on the **TIME** command. *Do not* specify a value of *NSTEP* = 0.

DT

Time interval at which output is written to the results file (**Jobname.RST**). If *NSTEP* is input, *DT* is ignored.

Command Default

Output will be written to **Jobname.RST** at 100 steps over the analysis.

Notes

You can use *NSTEP* or *DT* to specify the output interval to be used for **Jobname.RST**. You should not specify both quantities; if both are input, *NSTEP* will be used.

In an explicit dynamic small restart (**EDSTART**,2) or full restart analysis (**EDSTART**,3), the **EDRST** setting will default to the *NSTEP* or *DT* value used in the original analysis. You can issue **EDRST** in the restart to change this setting.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Output Controls>File Output Freq>Number of Steps
Main Menu>Solution>Output Controls>File Output Freq>Time Step Size

---

EDRUN,*Option*, *Cons*

Specify LS-DYNA serial or parallel processing.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> <>

Option
LS-DYNA processing option

SERIAL
Use serial processing (default)

SMP
Use Shared Memory Parallel processing

Cons
Consistency setting (only applicable when *Option* = SMP)

0
Result consistency is not required (default)

1
Result consistency is required

Command Default

The command default is serial processing.

Notes

You use this command to specify either serial (one CPU) processing or shared (multiple CPU) memory parallel processing (SMP). When you are using shared memory parallel processing, the calculations may be executed in different order, depending on CPU availability and the workload on each CPU. Because of this, you may see slight differences in the results when you run the same job multiple times, either with the same number or a different number of processors. Comparing nodal accelerations often shows wider discrepancies. To avoid these differences, you can specify that consistency be maintained by setting *Cons* = 1. Maintaining consistency can result in an increase of up to 15% in CPU time.

The parallel processing setting is only effective when you have multiple CPUs AND licenses for the appropriate number of ANSYS LS-DYNA SMP tasks. If you do not meet both of these requirements, this command will execute serial processing, regardless of command settings.

For more information on using SMP, see Solution Features in the *ANSYS/LS-DYNA User's Guide*.

Menu Paths

Main Menu>Solution>Analysis Options>Parallel Option

EDSHELL, *WPAN*, *SHNU*, *SHTC*, *WPBT*, *SHPL*, *ITRST*

Specifies shell computation controls for an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

WPAN

Maximum shell element warpage angle in degrees. Defaults to 20.

SHNU

Hughes-Liu shell normal update option:

2

Unique nodal fibers. This option is required for SHELL163 (KEYOPT(1) = 1, 6, or 7) if the real constant NLOC = 1 or -1.

1

Compute normals each cycle (default). This option is recommended.

1

Compute on restarts.

n

Compute every *n*th substep.

SHTC

Shell thickness change option:

0

No change.

1

Membrane straining causes thickness change. Important in sheet metal forming (default).

WPBT

Warping stiffness option for Belytschko-Tsay shells:

1

Belytschko-Wong-Chiang warping stiffness added. This option is recommended.

2

Belytschko-Tsay warping stiffness (default).

SHPL

Shell plasticity option. This option is only valid for these material models: strain rate independent plastic kinematic, strain rate dependent plasticity, power law plasticity, and piecewise linear plasticity.

1

Iterative plasticity with 3 secant iterations (default).

2

Full iterative plasticity.

3

Radial return noniterative plasticity. (Use this option with caution; it may lead to inaccurate results.)

ITRST
Triangular shell sorting option. If sorting is on, degenerate quadrilateral shell elements are treated as triangular shells.

1
Full sorting (default).

2
No sorting.

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Shell Elem Ctrls

EDSOLV
Specifies "explicit dynamics solution" as the subsequent status topic.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

This command is also valid in PREP7.

Menu Paths

Utility Menu>List>Status>LS-DYNA

EDSP, *Option*, *MIN*, *MAX*, *INC*
Specifies small penetration checking for contact entities in an explicit dynamic analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed (no default).

ON
Turn small penetration checking on for specified contact entities.

OFF
Turn small penetration checking off for specified contact entities.

LIST
List current setting for penetration checking.

MIN
Minimum contact entity number for which to turn on/off small penetration check (default = 1).

MAX
Maximum contact entity number for which to turn on/off small penetration check (defaults to *MIN*).

INC
Contact entity number increment (default = 1).

Command Default

Penetration checking is determined by *PENCHK* on the **EDCONTACT** command.

Notes

This command controls small penetration checking in an explicit dynamic analysis. **EDSP** is applicable only to the following contact types: STS, NTS, OSTS, TNTS, and TSTS. The penetration checking specified by **EDSP** is similar to *PENCHK* on the **EDCONTACT** command. However, **EDSP** controls penetration checking for individual contact entities whereas *PENCHK* is a global control that applies to all defined contact (of the types mentioned above). **EDSP** can be used in a new analysis, or in a small restart (**EDSTART**,2).

Use the **EDCLIST** command to list the contact entity numbers for all defined contact.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Contact>Advanced Controls

---

EDSTART, *RESTART*, *MEMORY*, *FSIZE*, *Dumpfile*

Specifies status (new or restart) of an explicit dynamics analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

RESTART
Status of the analysis (new or restart).

0
New analysis (default).

1
Simple restart.

2
Small restart.

3
Full restart.

MEMORY
Memory to be used (in words). If blank, LS-DYNA assigns a value (default). If more or less memory is needed, specify the number of words (a word is usually 32 bits on a workstation).

FSIZE
Scale factor for binary file sizes. Defaults to 7, which is (7x262144) = 1835008 words.

Dumpfile
Name of dump file to use during a restart (for example, **d3dump*nn***, where *nn* = 01, 02, 03,...,99 and defaults to 01). Leave this field blank when running a new analysis (*RESTART* = 0) so that the default dump file **d3dump01** will be created.

Command Default

A new analysis is assumed.

Notes

EDSTART can be issued before the **SOLVE** command to specify a new analysis, a simple restart, a small restart, or a full restart as described below.

New analysis: For a new analysis, you do not need to issue **EDSTART** unless you want to change the *MEMORY* or *FSIZE* option. If you do not specify the dump file name, **d3dump01** will be created by default.

Simple restart: This option assumes that the database has not been altered. Upon restarting, results will be appended to the existing results files. Issue **EDSTART**,1,,,**d3dump*nn*** to indicate which restart file to use as a starting point. The dump file to be used must have been created in an earlier run and must be available at the time this command is issued. You would typically use a simple restart when you interrupt the LS-DYNA run via CTRL-C and terminate the run prematurely by issuing the "sense switch control" key SW1 (see Solution Control and Monitoring in the *ANSYS LS-DYNA User's Guide*). At this point you should be able to view the partial solution using ANSYS postprocessors. After you are done viewing the partial solution, you can reenter the solution processor and issue **EDSTART**,1,,,**d3dump*nn***, followed by **SOLVE** to continue with the analysis. The results will be appended to the results files **Jobname.RST** and **Jobname.HIS**. You can perform multiple simple restarts by issuing **EDSTART**,1,,,**d3dump*nn*** multiple times, as needed. The solutions in the **Jobname.RST** file will all be in load step number 1.

Small restart: This option can be used when minor changes in the database are necessary. For example, you can reset the termination time, reset the output interval, add displacement constraints, change initial velocities, switch parts from a deformable to rigid state, etc. (See A Small Restart in the*ANSYS LS-DYNA User's Guide* for a complete description of database items that can be changed.) Issue **EDSTART**,2,,,**d3dump*nn*** followed by the commands required to change the database, then issue **SOLVE**. The results will be appended to the results files **Jobname.RST** and **Jobname.HIS**. You can perform multiple restarts by issuing **EDSTART**,2,,,**d3dump*nn*** multiple times, as needed. The additional restart solutions will be stored in **Jobname.RST** as load step numbers 2, 3, etc.

Full restart: A full restart is appropriate when many modifications to the database are required. For example, you can change the model geometry, apply different loading conditions, etc. Issue **EDSTART**,3,,,**d3dump*nn*** to denote a full restart analysis. The Jobname will automatically be changed to **Jobname_*nn***, (*nn* = 01 initially, and will be incremented each time **EDSTART**,3 is issued for subsequent full restarts). After the **EDSTART** command, you can input any commands needed to change the database. (Most commands which are applicable to an ANSYS LS-DYNA new analysis are also applicable to full restart analysis. A few commands related to contact specifications,

initial velocity definitions, and adaptive meshing are not supported.) Then issue the **EDIS** command to specify which portions of the model should be initialized in the full restart using results data from the **d3dump*nn*** file. Finally, issue the **SOLVE** command. At this point, new results files, **Jobname_*nn*.RST** and **Jobname_*nn*.HIS**, will be created. Time and output intervals in the new results files are continuous from the previous results files; the time is not reset to zero. (See A Full Restart in the *ANSYS LS-DYNA User's Guide* for a complete description of a full restart analysis.)

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Analysis Options>Restart Option

EDTERM, *Option*, *Lab*, *NUM*, *STOP*, *MAXC*, *MINC*

Specifies termination criteria for an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option

Label identifying the option to be performed.

ADD
: Define termination criteria (default).

DELE
: Delete termination criteria.

LIST
: List termination criteria.

Lab

Label identifying the type of termination (no default).

NODE
: Terminate solution based on nodal point coordinates. The analysis terminates when the current position of the specified node reaches either the maximum or minimum coordinate value (*STOP* = 1, 2, or 3), or when the node picks up force from any contact surface (*STOP* = 4).

PART
: Terminate solution based on rigid body (part) displacements. The analysis terminates when the displacement of the center of mass of the specified rigid body reaches either the maximum or minimum value (*STOP* = 1, 2, or 3), or when the displacement magnitude of the center of mass is exceeded (*STOP* = 4).

NUM

Node number (if *Lab* = NODE) or rigid body Part ID (if *Lab* = PART). (No default.)

STOP

Criterion for stopping the solution (no default).

1
: Global X-direction.

2
Global Y-direction.

3
Global Z-direction.

4
For *Lab* = NODE, stop the solution if contact occurs. For *Lab* = PART, stop the solution if the displacement magnitude is exceeded for the specified rigid body (use *MAXC* to define the displacement magnitude).

MAXC
Maximum (most positive) coordinate value (*Lab* = NODE) or displacement (*Lab* = PART). *MAXC* defaults to 1.0e21

MINC
Minimum (most negative) coordinate value (*Lab* = NODE) or displacement (*Lab* = PART). *MINC* defaults to -1.0e21.

Command Default

No termination criteria are defined other than the termination time set on the **TIME** command.

Notes

You may specify multiple termination criteria using **EDTERM**; the solution will terminate when any one of the criteria is satisfied, or when the solution end time (specified on the **TIME** command) is reached.

In an explicit dynamic small restart analysis (**EDSTART**,2) or full restart analysis (**EDSTART**,3), the termination criteria set in the previous analysis (the original analysis or the previous restart) are carried over to the restart. If the previous analysis terminated due to one of these criteria, that specific criterion must be modified so that it will not cause the restart to terminate prematurely. In particular, if a termination condition based on nodal contact (*Lab* = NODE, *STOP* = 4) is satisfied, this condition must be deleted and replaced with a condition based on nodal coordinates for that same node. (If a condition based on nodal coordinates already exists for that node, the replacement is not necessary.) In the restart, the number of termination criteria specified using **EDTERM** cannot exceed a maximum of 10 or the number specified in the original analysis.

Note that the termination criteria set by **EDTERM** are not active during dynamic relaxation (**EDDRELAX**).

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Analysis Options>Criteria to Stop>List
Main Menu>Solution>Analysis Options>Criteria to Stop>On a Node
Main Menu>Solution>Analysis Options>Criteria to Stop>On a Part

EDTP, *OPTION*, *VALUE1*, *VALUE2*

Plots explicit elements based on their time step size.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

OPTION

Plotting option (default = 1).

1

Plots the *elements* with the smallest time step sizes. The number of elements plotted and listed is equal to *VALUE1* (which defaults to 100). Each element is shaded red or yellow based on its time step value (see Notes for details).

2

Produces the same plot as for *OPTION* = 1, and also produces a list of the plotted elements and their corresponding time step values.

3

Produces a plot similar to *OPTION* = 1, except that all selected elements are plotted. Elements beyond the first *VALUE1* elements are blue and translucent. The amount of translucency is specified by *VALUE2*. This option also produces a list of the first *VALUE1* elements with their corresponding time step values.

VALUE1

Number of elements to be plotted and listed (default = 100). For example, if *VALUE1* = 10, only the elements with the 10 smallest time step sizes are plotted and listed.

VALUE2

Translucency level ranging from 0 to 1 (default = 0.9). *VALUE2* is only used when *OPTION* = 3, and only for the elements plotted in blue. To plot these elements as non-translucent, set *VALUE2* = 0.

Notes

EDTP invokes an ANSYS macro that plots and lists explicit elements based on their time step size. For *OPTION* = 1 or 2, the number of elements plotted is equal to *VALUE1* (default = 100). For *OPTION* = 3, all selected elements are plotted.

The elements are shaded red, yellow, or blue based on their time step size. Red represents the smallest time step sizes, yellow represents the intermediate time step sizes, and blue represents the largest time step sizes. For example, if you specify *VALUE1* = 30, and if T1 is the smallest critical time step of all elements and T30 is the time step of the 30th smallest element, then the elements are shaded as follows:

Red - time step range is T1 to T1 + [0.05*(T30--T1)]
Yellow - time step range is T1 + [0.05*(T30--T1)] to T30
Blue (translucent) - time step range is > T30

Translucent blue elements only appear when OPTION = 3.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Time Controls>Time Step Prediction

EDVEL, *Option*, *Cname*, *VX*, *VY*, *VZ*, *OMEGAX*, *OMEGAY*, *OMEGAZ*, *XC*, *YC*, *ZC*, *ANGX*, *ANGY*, *ANGZ*

Applies initial velocities to nodes or node components in an explicit dynamic analysis.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed.

VGEN
Define initial velocities based on translational velocities (relative to global Cartesian) and the rotational velocity about an arbitrary axis. For this option, use the fields *VX*, *VY*, *VZ* to specify the translational velocities, and use *OMEGAX*, *XC*, *YC*, *ZC*, *ANGX*, *ANGY*, *ANGZ* to specify the rotational velocity and the axis of rotation.

VELO
Define initial velocity based on translational velocities and nodal rotational velocities input relative to the global Cartesian axes. For this option, use the following fields to define the initial velocity: *VX*, *VY*, *VZ*, *OMEGAX*, *OMEGAY*, *OMEGAZ*.

LIST
List initial velocity for the component or node specified by *Cname*. If *Cname* is blank, all initial velocities defined on nodes and node components are listed. Remaining fields are ignored for this option.

DELE
Delete initial velocity defined for the component or node specified by *Cname*. If *Cname* is blank, all initial velocities defined on nodes and node components are deleted. Remaining fields are ignored for this option.

Cname
Name of existing component [**CM**] or node number to which the initial velocity is to be applied. If a component is used, it must consist of nodes.

VX
Initial velocity in X direction. Defaults to 0.

VY
Initial velocity in Y direction. Defaults to 0.

VZ
Initial velocity in Z direction. Defaults to 0.

OMEGAX
For *Option* = VGEN, *OMEGAX* is the initial rotational velocity of the component (or node) about the specified rotational axis. For *Option* = VELO, *OMEGAX* is the initial nodal rotational velocity about the X-axis. *OMEGAX* defaults to 0.

OMEGAY
Initial nodal rotational velocity about the Y-axis (used only if *Option* = VELO). Defaults to 0.

OMEGAZ
Initial nodal rotational velocity about the Z-axis (used only if *Option* = VELO). Defaults to 0.

The remaining fields are used only if *Option* = VGEN.

XC
X coordinate on rotational axis. Defaults to 0.

YC
Y coordinate on rotational axis. Defaults to 0.

ZC
Z coordinate on rotational axis. Defaults to 0.

ANGX
Angle relative to global X-axis. Defaults to 0.

ANGY
Angle relative to global Y-axis. Defaults to 0.

ANGZ
Angle relative to global Z-axis. Defaults to 0.

Notes

You cannot mix the two methods of initial velocity input (*Option* = VELO and *Option* = VGEN) in the same analysis. You must use only one method for all initial velocity definitions.

The VGEN and VELO methods differ in how the rotational velocity is defined. Use *Option* = VGEN to input the initial velocities of a rotating component. Use *Option* = VELO to apply the rotations directly to the nodes' rotation degrees of freedom. Since only shell and beam elements have rotation degrees of freedom, the rotations input with *Option* = VELO are only applicable to SHELL163 and BEAM161 elements. The rotational velocities input with *Option* = VELO are ignored for nodes not having rotational degrees of freedom (such as nodes attached to a SOLID164 or SOLID168 element).

It is normally acceptable to mix nodes belonging to deformable bodies and rigid bodies in the nodal component used in an initial velocity definition. However, when defining initial velocities in an implicit-to-explicit sequential solution, this is not an acceptable practice. In order for the initial velocities to be defined correctly in this type of analysis, you must define the initial velocities on the deformable body nodes separately from the initial velocities on the rigid body nodes.

Issuing the **EDVEL** command again for the same component or node (*Cname*) will overwrite previous initial velocities defined for that component or node.

To set the initial velocities to zero, issue the **EDVEL** command with only the *Option* (use VELO or VGEN) and *Cname* fields specified.

In a small restart analysis (**EDSTART**,2), you can only use the *Option* = VELO method to change initial velocities. When used in a small restart, the command **EDVEL**,VELO changes the velocity of the specified nodes. If you don't change the velocity of the nodes, their velocity at the beginning of the restart will be the same as the velocity at the end of the previous analysis.

Except for the LIST option, the **EDVEL** command is not supported in a full restart analysis (**EDSTART**,3). You can list initial velocities defined in the previous analysis with the command **EDVEL**,LIST. However, you cannot change initial velocities for nodes or node components that existed in the previous analysis; their velocity at the beginning of the analysis will be the same as the velocity at the end of the previous analysis. In addition, you cannot define initial velocities for any nodes that are added in the full restart; the velocity of new nodes will be zero.

To apply initial velocities to parts or part assemblies, use the **EDPVEL** command.

You can use **EDPVEL** and **EDVEL** in the same analysis. If a node or node component input on the **EDVEL** command shares common nodes with a part or part assembly input on the **EDPVEL** command, the initial velocities defined on the common nodes will be determined by the last command input.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Nodes>Delete
Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Nodes>List
Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Nodes>w/Axial Rotate
Main Menu>Preprocessor>LS-DYNA Options>Initial Velocity>On Nodes>w/Nodal Rotate
Main Menu>Solution>Initial Velocity>On Nodes>Delete
Main Menu>Solution>Initial Velocity>On Nodes>List
Main Menu>Solution>Initial Velocity>On Nodes>w/Axial Rotate
Main Menu>Solution>Initial Velocity>On Nodes>w/Nodal Rotate

EDWELD, *Option*, *NWELD*, *N1*, *N2*, *SN*, *SS*, *EXPN*, *EXPS*, *EPSF*, *TFAIL*, *NSW*, *CID*

Defines a massless spotweld or generalized weld for use in an explicit dynamic analysis.

PREP7: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Label identifying the option to be performed:

ADD
Define a weld (default). This weld may be a spotweld between two nodes or a generalized weld. A massless spotweld will be defined if valid node numbers are specified in fields *N1* and *N2*. A generalized weld will be defined if a node component is specified in field *N1*.

DELE
Delete specified weld. If *NWELD* is blank, all welds are deleted.

LIST
List specified weld. If *NWELD* is blank, all welds are listed.

NWELD
Reference number identifying the spotweld or generalized weld.

N1, *N2*
For a spotweld, *N1* and *N2* are the nodes which are connected by the spotweld. For a generalized weld, input a nodal component name in *N1* and leave *N2* blank. The nodal component should contain all nodes that are to be included in the generalized weld.

SN
Normal force at spotweld failure.

SS
Shear force at spotweld failure.

EXPN
Exponent for normal spotweld force.

EXPS
Exponent for shear spotweld force.

EPSF
Effective plastic strain at ductile failure (used only for a generalized weld).

TFAIL
Failure time for constraint set (used only for a generalized weld); default = 1.0e20.

NSW
Number of spot welds for the generalized weld.

CID
Coordinate system ID number (CID) to be used for output data (used only for a generalized weld). The coordinate system must be previously defined with the **EDLCS** command.

Notes

This command can be used to define a massless spotweld between two nodes or a generalized weld for a group of nodes. For a spotweld, the nodes specified by *N1* and *N2* must not be coincident. For a generalized weld, coincident nodes are permitted, but *CID* must be specified when using coincident nodes. **EDWELD** is not updated after a node merge operation; therefore, node merging [**NUMMRG**,NODE] should be done before any **EDWELD** definitions. Nodes connected by a spotweld or generalized weld cannot be constrained in any other way.

Failure of the weld occurs when:

$$\left(\frac{|f_n|}{S_n}\right)^{expn} + \left(\frac{|f_s|}{S_s}\right)^{exps} \geq 1$$

where f_n and f_s are normal and shear interface forces. Normal interface force f_n is nonzero for tensile values only.

You can graphically display spotwelds by issuing the command **/PBC**,WELD,,1.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Spotweld>Delete
Main Menu>Preprocessor>LS-DYNA Options>Spotweld>Genrlizd Spotwld
Main Menu>Preprocessor>LS-DYNA Options>Spotweld>List
Main Menu>Preprocessor>LS-DYNA Options>Spotweld>Massless Spotwld

---

EDWRITE, *Option*, *Fname*, *Ext*, --

Writes explicit dynamics input to an LS-DYNA input file.

SOLUTION: Explicit Dynamics

<> <> <> DY <> <> <> <> <> <> ED

Option
Sets a flag in the LS-DYNA input file (***Fname.Ext***) to produce desired output.

ANSYS
Set a flag to write results files for the ANSYS postprocessors (default). The files that will be written are **Jobname.RST** and **Jobname.HIS** (see Notes below).

LSDYNA
Set a flag to write results files for the LS-DYNA postprocessor (LS-POST). The files that will be written are **D3PLOT**, and files specified by **EDOUT** and **EDHIST** (see Notes below).

BOTH
Set a flag to write results files for both ANSYS and LS-DYNA postprocessors.

Fname
File name and directory path (80 characters maximum, including directory; this limit is due to an LS-DYNA program limitation). If you do not specify a directory path, it will default to your working directory. The file name defaults to **Jobname**. Previous data on this file, if any, are overwritten.

Ext
Filename extension (8 character maximum).

The extension defaults to K in a new analysis and in a full restart analysis, and to R in a small restart analysis.

--
Unused field

Notes

This command writes an LS-DYNA input file for the LS-DYNA solver. **EDWRITE** is only valid if explicit dynamic elements have been specified. This command is not necessary if the LS-DYNA solver is invoked from within ANSYS, in which case **Jobname.K** (or **Jobname.R**) is written automatically when the solution is initiated. (If LS-DYNA is invoked from within ANSYS, use **EDOPT** to specify desired output.)

If the analysis is a small restart (**EDSTART**,2), the file that is written will have the name **Jobname.R** (by default) and will only contain changes from the original analysis.

If the analysis is a full restart (**EDSTART**,3), the file that is written will have the name **Jobname_*nn*.K** (by default) and will contain all the information from the database. In a full restart, the jobname is changed to **Jobname_*nn*** (*nn* = 01 initially, and is incremented for each subsequent full restart.)

A command is included in the LS-DYNA input file to instruct the LS-DYNA solver to write the results files indicated by *Option*. By default, LS-DYNA will write the ANSYS results file **Jobname.RST** (see the **EDRST** command). If **Jobname.HIS** is desired, you must also issue **EDHIST**.

Option = LSDYNA or BOTH will cause LS-DYNA to write results files for the LS-POST postprocessor. The **D3PLOT** file is always written for these two options. If other LS-POST files are desired, you must issue the appropriate **EDHIST** and **EDOUT** commands.

This command is also valid in PREP7.

Menu Paths

Main Menu>Solution>Write Jobname.k

/EFACET, *NUM*

Specifies the number of facets per element edge for PowerGraphics displays.

POST1: Controls

MP ME ST DY <> PR EM <> FL PP ED

NUM
Number of facets per element edge for element plots.

1
Use 1 facet per edge (default for h-elements).

2
Use 2 facets per edge (default for p-elements).

4
Use 4 facets per edge.

Command Default

As stated above.

Notes

/EFACET is valid only when PowerGraphics is enabled [**/GRAPHICS**,POWER], except that it can be used in FULL graphics mode for element CONTA174. (See the **/GRAPHICS** command and element CONTA174 in the *ANSYS Elements Reference* for more information.) The **/EFACET** command is only applicable to element type displays.

/EFACET controls the fineness of the subgrid that is used for element plots. The element is subdivided into smaller portions called *facets*. Facets are piecewise linear surface approximations of the actual element face. In their most general form, facets are warped planes in 3-D space. A greater number of facets will result in a smoother representation of the element surface for element plots. **/EFACET** may affect results averaging. See Contour Displays in the *ANSYS Basic Analysis Guide* for more information.

For midside node elements, use *NUM* = 2; if *NUM* = 1, no midside node information is output. For non-midside node elements, *NUM* should be set to 1. See the **PLNSOL** and **PRNSOL** commands for more information.

With PowerGraphics active (**/GRAPHICS**,POWER), the averaging scheme for surface data with interior element data included (**AVRES**,,FULL) and multiple facets per edge (**/EFACET**,2 or **/EFACET**,4) will yield differing minimum and maximum contour values depending on the Z-Buffering options (**/TYPE**,,6 or **/TYPE**,,7). When the Section data is not included in the averaging schemes (**/TYPE**,,7), the resulting absolute value for the midside node is significantly smaller.

Caution: *If you specify* **/EFACET**,1, *PowerGraphics does not plot midside nodes. You must use* **/EFACET**,2 *to make the nodes visible.*

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Options for Outp
Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu
Utility Menu>List>Results>Options
Utility Menu>Plot>Results>Contour Plot>Nodal Solution
Utility Menu>PlotCtrls>Multi-Plot Contrls
Utility Menu>PlotCtrls>Style>Size and Shape

EGEN, *ITIME*, *NINC*, *IEL1*, *IEL2*, *IEINC*, *MINC*, *TINC*, *RINC*, *CINC*, *SINC*, *DX*, *DY*, *DZ*

Generates elements from an existing pattern.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

ITIME, *NINC*

Do this generation operation a total of *ITIME*s, incrementing all nodes in the given pattern by *NINC* each time after the first. *ITIME* must be >1 if generation is to occur. *NINC* may be positive, zero, or negative. If *DX*, *DY*, and/or *DZ* is specified, *NINC* should be set so any existing nodes (as on **NGEN**) are not overwritten.

IEL1, *IEL2*, *IEINC*

Generate elements from selected pattern beginning with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* is negative, *IEL2* and *IEINC* are ignored and the last |*IEL1*| elements (in sequence backward from the maximum element number) are used as the pattern to be repeated. If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and use all selected elements [**ESEL**] as pattern to be repeated. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *INC* are ignored).

MINC

Increment material number of all elements in the given pattern by *MINC* each time after the first.

TINC

Increment type number by *TINC*.

RINC

Increment real constant table number by *RINC*.

CINC

Increment element coordinate system number by *CINC*.

SINC

Increment section ID number by *SINC*.

DX, *DY*, *DZ*

Define nodes that do not already exist but are needed by generated elements (as though the **NGEN**,*ITIME*,*INC*,*NODE1*,,,*DX*,*DY*,*DZ* were issued before **EGEN**). Zero is a valid value. If blank, *DX*, *DY*, and *DZ* are ignored.

Notes

A pattern may consist of any number of previously defined elements. The MAT, TYPE, REAL, ESYS, and SECNUM numbers of the new elements are based upon the elements in the pattern and not upon the current specification settings.

You can use the **EGEN** command to generate interface elements (INTER192, INTER193, INTER194, and INTER195) directly. However, because interface elements require that the element connectivity be started from the bottom surface, you must make sure that you use the correct element node connectivity. See the element descriptions for INTER192, INTER193, INTER194, and INTER195 for the correct element node definition.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Elements>Auto Numbered

EINTF, *TOLER*, *K*, *TLAB*, *KCN*, *DX*, *DY*, *DZ*, *KNONROT*

Defines two-node elements between coincident or offset nodes.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

TOLER

Tolerance for coincidence (based on maximum Cartesian coordinate difference for node locations and on angle differences for node orientations). Defaults to 0.0001. Only nodes within the tolerance are considered to be coincident.

K

Only used when the type of the elements to be generated is PRETS179. *K* is the pretension node that is common to the pretension section that is being created. If *K* is not specified, it will be created by ANSYS automatically and will have an ANSYS-assigned node number. If *K* is specified but does not already exist, it will be created automatically but will have the user-specified node number. *K* cannot be connected to any existing element.

TLAB

Nodal number ordering. Allowable values are:

LOW

The 2-node elements are generated from the lowest numbered node to the highest numbered node.

HIGH

The 2-node elements are generated from the highest numbered node to the lowest numbered node.

REVE

Reverses the orientation of the selected 2-node element.

KCN

In coordinate system *KCN*, elements are created between node 1 and node 2 (= node 1 + dx dy dz)..

DX, *DY*, *DZ*

Node location increments that define the node offset in the active coordinate system (DR, Dθ, DZ for cylindrical and DR, Dθ, DΦ for spherical or toroidal).

KNONROT

When *KNONROT* = 0, the nodes belonging to the elements created are rotated into coordinate system *KCN* (see **NROTATE** command description). When *KNONROT* = 1, the nodes coordinate system is not rotated.

Notes

Defines 2-node elements (such as gap elements) between coincident or offset nodes (within a tolerance). May be used, for example, to "hook" together elements interfacing at a seam, where the seam consists of a series of node pairs. One element is generated for each set of two coincident nodes. For more than two coincident or offset nodes in a cluster, an element is generated from the lowest numbered node to each of the other nodes in the cluster. If fewer than all nodes are to be checked for coincidence, use the **NSEL** command to select the nodes. Element numbers are incremented by one from the highest previous element number. The element type must be set [**ET**] to a 2-node element before issuing this command. Use the **CPINTF** command to connect nodes by coupling instead of by elements. Use the **CEINTF** command to connect the nodes by constraint equations instead of by elements.

For contact element CONTA178, the tolerance is based on the maximum Cartesian coordinate difference for node locations only. The angle differences for node orientations are not checked.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Auto Numbered>At Coincid Nd
Main Menu>Preprocessor>Modeling>Create>Elements>Auto Numbered>Offset Nodes

---

EKILL, *ELEM*

Deactivates an element (for the birth and death capability).

SOLUTION: Birth and Death

MP ME ST <> <> <> <> <> <> PP ED

ELEM

Element to be deactivated. If ALL, deactivate all selected elements [**ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *ELEM*.

Notes

Deactivates the specified element when the birth and death capability is being used. A deactivated element remains in the model but contributes a near-zero stiffness (or conductivity etc.) value (see the **ESTIF** command) to the overall matrix. Deactivated elements contribute nothing to the overall mass (or capacitance, etc.) matrix. The element may be reactivated with the **EALIVE** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Birth & Death>Kill Elements
Main Menu>Solution>Load Step Opts>Other>Birth & Death>Kill Elements

---

ELEM

Specifies "Elements" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Elements

ELIST, *IEL1*, *IEL2*, *INC*, *NNKEY*, *RKEY*, *PTKEY*

Lists the elements and their attributes.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

IEL1, *IEL2*, *INC*

Lists elements from *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *INC* (defaults to 1). If *IEL1* = ALL (default), *IEL2* and *INC* are ignored and all selected elements [**ESEL**] are listed. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *INC* are ignored).

NNKEY

Node listing key:

0
: List attribute references and nodes.

1
: List attribute references but not nodes.

RKEY

Real constant listing key:

0
: Do not show real constants for each element.

1
: Show real constants for each element. This includes default values chosen for the element.

PTKEY

LS-DYNA part number listing key (applicable to ANSYS LS-DYNA only):

0
: Do not show part ID number for each element.

1
: Show part ID number for each element.

Notes

Lists the elements with their nodes and attributes (MAT, TYPE, REAL, ESYS, SECNUM, PART). See also the **LAYLIST** command for listing layered elements.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Elements>Attributes + RealConst
Utility Menu>List>Elements>Attributes Only
Utility Menu>List>Elements>Nodes + Attributes
Utility Menu>List>Elements>Nodes + Attributes + RealConst

EMAGERR

Calculates the relative error in an electrostatic or electromagnetic field analysis.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

Notes

The relative error is an approximation of the mesh discretization error associated with a solution. It is based on the discrepancy between the unaveraged, element-nodal field values and the averaged, nodal field values. The calculation is valid within a material boundary and does not consider the error in continuity of fields across dissimilar materials.

For electrostatics, the field values evaluated are the electric field strength (EFSUM) and the electric flux density (DSUM). A relative error norm of each is calculated on a per-element basis and stored in the element table [**ETABLE**] with the labels EF_ERR and D_ERR. Normalized error values EFN_ERR and DN_ERR are also calculated and stored in the element table. Corresponding quantities for electromagnetics are H_ERR, B_ERR, HN_ERR, and BN_ERR, which are calculated from the magnetic field intensity (HSUM) and the magnetic flux density (BSUM). The normalized error value is the relative error norm value divided by the peak element-nodal field value for the currently selected elements.

Use the **PLETAB** and **PRETAB** commands to plot and list the error norms and normalized error values.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Element Based>Error Eval

---

EMATWRITE, *Key*

Forces the writing of all the element matrices to File.EMAT.

SOLUTION: Analysis Options

MP ME ST <> <> <> <> <> <> PP ED

Key

Write key:

YES
: Forces the writing of the element matrices to **File.EMAT** even if not normally done.

NO
: Element matrices are written only if required (default).

Notes

Forces the program to write the **File.EMAT** and is used if you intend to issue a subsequent **PSOLVE** that requires **File.EMAT** as a prerequisite or if you plan to follow the initial load step with a subsequent inertia relief calculation using the **IRLF** command.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

EMF

Calculates the electromotive force (emf), or voltage drop along a predefined path.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

Notes

EMF invokes an ANSYS macro which calculates the electromotive force (emf), or voltage drop along a predefined path (specified with the **PATH** command). It is valid for both 2-D and 3-D electric field analysis or high-frequency electromagnetic field analysis. The calculated emf value is stored in the parameter EMF.

You must define a line path (via the **PATH** command) before issuing the **EMF** command macro. The macro uses calculated values of the electric field (EF), and uses path operations for the calculations. All path items are cleared when the macro finishes executing.

The **EMF** macro sets the "ACCURATE" mapping method and "MAT" discontinuity option on the **PMAP** command. The ANSYS program retains these settings after executing the macro.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>EMF

EMID, *Key*, *Edges*

Adds or removes midside nodes.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

Key

Add or remove key:

ADD
: Add midside node to elements (default).

REMOVE
: Remove midside nodes from elements.

Edges

ALL
: Add (or remove) midside nodes to (from) all edges of all selected elements, independent of which nodes are selected (default).

EITHER
: Add (or remove) midside nodes only to (from) element edges which have either corner node selected.

BOTH
Add (or remove) midside nodes only to (from) element edges which have both corner nodes selected.

Notes

This command adds midside nodes to (or removes midside nodes from) the selected elements. For this to occur, the selected elements must be midside node capable, the active element type [**TYPE**] must allow midside node capability, and the relationship between the finite element model and the solid model (if any) must first be disassociated [**MODMSH**].

By default, **EMID** generates a midside node wherever a zero (or missing) midside node occurs for that element. You can control this and add (or remove) midside nodes selectively by using the *Edges* argument. Nodes are located midway between the two appropriate corner nodes based on a linear Cartesian interpolation. Nodal coordinate system rotation angles are also linearly interpolated. Connected elements share the same midside node. Node numbers are generated sequentially from the maximum node number.

The **EMID** command is useful for transforming linear element types to quadratic element types having the same corner node connectivity (for example, by changing the element type from PLANE42 to PLANE82 [**ET**] and then issuing the **EMID** command). **EMID** is also useful for transforming elements created outside of the ANSYS program.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Add Mid Nodes
Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Remove Mid Nd

---

EMIS, *MAT*, *EVALU*

Specifies emissivity as a material property for the Radiation Matrix method.

AUX12: Radiation Substructures

MP ME <> <> <> PR <> <> <> PP ED

MAT
Material number associated with this emissivity (500 maximum). Defaults to 1.

EVALU
Emissivity for this material ($0.0 < EVALU \leq 1.0$). Enter a very small number for zero.

Command Default

Emissivity value of 1.0 is associated with all materials.

Notes

Specifies emissivity as a material property for the Radiation Matrix method. This material property can then be associated with each element.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Emissivities

EMODIF, *IEL*, *STLOC*, *I1*, *I2*, *I3*, *I4*, *I5*, *I6*, *I7*, *I8*

Modifies a previously defined element.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

IEL
Modify nodes and/or attributes for element number *IEL*. If ALL, modify all selected elements [**ESEL**]. If *IEL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL*.

STLOC
Starting location (*n*) of first node to be modified or the attribute label. If *n*, modify element node positions *n*, *n*+1, etc. (*n* = 1 to 20). For example, if *STLOC* = 1, *I1* refers to the first node, *I2*, the second, etc. If *STLOC* = 9, *I1* refers to the ninth node, *I2*, the tenth, etc. Attributes are also modified to the currently specified values (use -*n* to modify only nodes and not attributes). If zero, modify only the attributes to the currently specified values. If MAT, TYPE, REAL, ESYS, or SECNUM, modify only that attribute to the *I1* value.

I1, *I2*, *I3*, *I4*, *I5*, *I6*, *I7*, *I8*
Replace the previous node numbers assigned to this element with these corresponding values. A (blank) retains the previous value (except in the *I1* field, which resets the *STLOC* node number to zero). For attributes, replace the existing value with the *I1* value (or the default if *I1* is zero or blank).

Notes

The nodes and/or attributes (MAT, TYPE, REAL, ESYS, and SECNUM values) of an existing element may be changed with this command.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Modify Attrib
Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Modify Nodes

---

EMORE, *Q*, *R*, *S*, *T*, *U*, *V*, *W*, *X*

Adds more nodes to the just-defined element.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

Q, *R*, *S*, *T*, *U*, *V*, *W*, *X*
Numbers of nodes typically assigned to ninth (node *Q*) through sixteenth (node *X*) nodal positions, if any. If *Q* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Repeat **EMORE** command for up to 4 additional nodes (20 maximum). Nodes are added after the last *nonzero* node of the element. Node numbers defined with this command may be zeroes.

Menu Paths

This command cannot be accessed from a menu.

EMSYM, *NSECT*

Specifies circular symmetry for electromagnetic sources.

PREP7: Special Purpose

MP ME ST DY <> <> EM <> <> PP ED

NSECT

The number of circular symmetry sections (defaults to 1).

Notes

Specifies the number of times to repeat electromagnetic sources for circular symmetry. Applies to SOURC36 elements and to coupled-field elements with electric current conduction results in the database. Sources are assumed to be equally spaced over 360° about the global Cartesian Z axis.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>Copy Sources
Main Menu>Solution>Load Step Opts>Magnetics>Options Only>Copy Sources

---

EMTGEN, *Ncomp*, *Ecomp*, *PNcomp*, *DOF*, *GAP*, *GAPMIN*, *FKN*, *PER0*

Generates a set of TRANS126 elements.

PREP7: Elements

MP <> <> <> <> <> <> <> <> PP ED

Ncomp

Component name of the surface nodes of a structure which attach to the TRANS126 elements. You must enclose name-strings in single quotes in the **EMTGEN** command line.

Ecomp

Component name of the TRANS126 elements generated. You must enclose name-strings in single quotes in the **EMTGEN** command line. Defaults to EMTELM.

PNcomp

Component name of the plane nodes generated by the command at an offset (*GAP*) from the surface nodes. You must enclose name-strings in single quotes in the **EMTGEN** command line. Defaults to EMTPNO.

DOF

Active structural degree of freedom (DOF) for TRANS126 elements (UX, UY, or UZ) in the Cartesian coordinate system. You must enclose the DOF in single quotes.

GAP

Initial gap distance from the surface nodes to the plane. Be sure to use the correct sign with respect to Ncomp node location.

GAPMIN

Minimum gap distance allowed (GAPMIN real constant) for TRANS126 elements. Defaults to the absolute value of (*GAP*)*0.05.

FKN
Contact stiffness factor used as a multiplier for a contact stiffness appropriate for bulk deformation. Defaults to 0.1.

PER0
Free-space permittivity. Defaults to 8.854e-6 (μMKS units).

Notes

The **EMTGEN** command generates a set of TRANS126 elements between the surface nodes of a moveable structure and a plane of nodes, typically representing a ground plane. The plane of nodes are created by the command at a specified offset (*GAP*). Each element attaches to a surface node and to a corresponding node representing the plane. The created elements are set to the augmented stiffness method (KEYOPT(6) = 1), which can help convergence. The generated plane nodes should be constrained appropriately for the analysis.

You can use TRANS126 elements for simulating fully coupled electrostatic structural coupling between a MEMS device and a plane, if the gap distance between the device and the plane is small compared to the overall surface area dimensions of the device. This assumption allows for a point-wise closed-form solution of capacitance between the surface nodes and the plane; i.e. CAP = PER0*AREA/GAP, where PER0 if the free-space permittivity, AREA is the area associated with the node, and GAP is the gap between the node and the plane. The area for each node is computed using the ARNODE function in ANSYS. See the ***GET** command description for more information on the ARNODE function.

With a distributed set of TRANS126 elements attached directly to the structure and a plane (such as a ground plane), you can perform a full range of coupled electrostatic-structural simulations, including:

- Static analysis (due to a DC voltage or a mechanical load)
- Prestressed modal analysis (eigenfrequencies, including frequency-shift effects of a DC bias voltage)
- Prestressed harmonic analysis (system response to a small-signal AC excitation with a DC bias voltage or mechanical load)
- Large signal transient analysis (time-transient solution due to an arbitrary time-varying voltage or mechanical excitation)

The TRANS126 element also employs a node-to-node gap feature so you can perform contact-type simulations where the structure contacts a plane (such as a ground plane). The contact stiffness factor, FKN, is used to control contact penetration once contact is initiated. A smaller value provides for easier convergence, but with more penetration.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Transducers>Electromechanic>Node to Plane

---

EMUNIT, *Lab*, *VALUE*

Specifies the system of units for magnetic field problems.

PREP7: Materials

MP <> <> <> <> <> EM <> <> PP ED

Lab
Label specifying the type of units:

MKS
Rationalized MKS system of units (meters, amperes, henries, webers, etc.). Free-space permeability is set to 4 πe-7 henries/meter. Free-space permittivity is set to 8.85 e-12 F/m.

MUZRO
User defined system of units. Free-space permeability is set to the value input for *VALUE*. Other units must correspond to the permeability units. Relative permeability may be altered to absolute values.

EPZRO
User defined system of units. Free-space permittivity is set to the value input for VALUE. Other units must correspond to the permittivity units.

VALUE
User value of free-space permeability (defaults to 1) if *Lab* = MUZRO, or free-space permittivity (defaults to 1) if *Lab* = EPZRO.

Command Default

Rationalized MKS system of units (meters, amperes, henries, webers, etc.). Free-space permeability is set to 4 πe-7 Henries/meter, free-space permittivity is set to 8.85e-12 Farads/meter.

Notes

Specifies the system of units to be used for electric and magnetic field problems. The free-space permeability and permittivity values may be set as desired. These values are used with the relative property values [**MP**] to establish absolute property values.

Note — If the magnetic source field strength (H_s) has already been calculated [**BIOT**], switching **EMUNIT** will not change the values.

For micro-electromechanical systems (MEMS), where dimensions are on the order of microns, see the conversion factors in System of Units in the *ANSYS Coupled-Field Analysis Guide*.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Electromag Units
Main Menu>Preprocessor>Material Props>Electromag Units
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Electromag Units

---

EN, *IEL*, *I*, *J*, *K*, *L*, *M*, *N*, *O*, *P*

Defines an element by its number and node connectivity.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

IEL
Number assigned to element being defined. If *I* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

I
Number of node assigned to first nodal position (node I).

J, *K*, *L*, *M*, *N*, *O*, *P*
Number assigned to second (node J) through eighth (node P) nodal position, if any.

Notes

Defines an element by its nodes and attribute values. Similar to the **E** command except it allows the element number (*IEL*) to be defined explicitly. Element numbers need not be consecutive. Any existing element already having this number will be redefined.

Up to 8 nodes may be specified with the **EN** command. If more nodes are needed for the element, use the **EMORE** command. The number of nodes required and the order in which they should be specified are described in the *ANSYS Elements Reference* for each element type. The current (or default) MAT, TYPE, REAL, SECNUM, and ESYS attribute values are also assigned to the element.

When creating elements with more than 8 nodes using this command and the **EMORE** command, it may be necessary to turn off shape checking using the **SHPP** command before issuing this command. If a valid element type can be created without using the additional nodes on the **EMORE** command, this command will create that element. The **EMORE** command will then modify the element to include the additional nodes. If shape checking is active, it will be performed before the **EMORE** command is issued. Therefore, if the shape checking limits are exceeded, element creation may fail before the **EMORE** command modifies the element into an acceptable shape.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>User Numbered>Thru Nodes

---

ENDRELEASE, --, *TOLERANCE*, *Dof1*, *Dof2*, *Dof3*, *Dof4*

Specifies degrees of freedom to be decoupled for end release.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

--
Unused field

TOLERANCE
Angle tolerance (in degrees) between adjacent elements. Defaults to 20°. Set *TOLERANCE* to -1 to indicate all selected elements.

Dof1, *Dof2*, *Dof3*, *Dof4*
Degrees of freedom to release. If *Dof1* is blank, WARP is assumed and *Dof2*, *Dof3*, and *Dof4* are ignored.

WARP
Release the warping degree of freedom (default).

ROTX
Release rotations in the X direction.

ROTY
Release rotations in the Y direction.

ROTZ
Release rotations in the Z direction.

UX
Release displacements in the X direction.

UY
Release displacements in the Y direction.

UZ
Release displacements in the Z direction.

BALL
Create ball joints (equivalent to releasing WARP, ROTX, ROTY, and ROTZ).

Notes

This command specifies end releases for BEAM188 and BEAM189 elements. The command works on currently selected nodes and elements. It creates end releases on any two connected beam elements whose angle at connection exceeds the specified tolerance. From within the GUI, the **Picked node** option generates an end release at the selected node regardless of the angle of connection (angle tolerance is set to -1).

Use the **CPLIST** command to list the coupled sets generated by the **ENDRELEASE** command.

Note — You should exercise due engineering judgement when using this command, as improper use may result in mechanics that render a solution impossible.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>End Releases>On Selected set
Main Menu>Solution>Load Step Opts>Other>End Releases>On Selected set

---

ENGEN, *IINC*, *ITIME*, *NINC*, *IEL1*, *IEL2*, *IEINC*, *MINC*, *TINC*, *RINC*, *CINC*, *SINC*, *DX*, *DY*, *DZ*

Generates elements from an existing pattern.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

IINC
Increment to be added to element numbers in pattern.

ITIME, *NINC*
Do this generation operation a total of *ITIME*s, incrementing all nodes in the given pattern by *NINC* each time after the first. *ITIME* must be > 1 if generation is to occur. *NINC* may be positive, zero, or negative.

IEL1, *IEL2*, *IEINC*
Generate elements from the pattern that begins with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* is negative, *IEL2* and *IEINC* are ignored and use the last |*IEL1*| elements (in sequence backward from the maximum element number) as the pattern to be repeated. If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and all selected elements [**ESEL**] are used as the pattern to be repeated. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *IEINC* are ignored).

MINC
Increment material number of all elements in the given pattern by *MINC* each time after the first.

TINC
Increment type number by *TINC*.

RINC
Increment real constant table number by *RINC*.

CINC
Increment element coordinate system number by *CINC*.

SINC
Increment section ID number by *SINC*.

DX, *DY*, *DZ*
Define nodes that do not already exist but are needed by generated elements (**NGEN**,*ITIME, INC, NODE1, , , DX, DY, DZ*). Zero is a valid value. If blank, *DX*, *DY*, and *DZ* are ignored.

Notes

Same as the **EGEN** command except it allows element numbers to be explicitly incremented (*IINC*) from the generated set. Any existing elements already having these numbers will be redefined.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Elements>User Numbered

ENORM, *ENUM*

Reorients shell element normals.

PREP7: Elements

MP ME ST DY <> PR <> <> <> PP ED

ENUM
Element number having the normal direction that the reoriented elements are to match. If *ENUM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Reorients shell elements so that their outward normals are consistent with that of a specified element. The operation reorients the element by reversing and shifting the node connectivity pattern. For example, for a 4-node shell element, the nodes in positions I, J, K and L of the original element are placed in positions J, I, L and K of the reoriented element. All 3-D shell elements in the selected set are considered for reorientation, and no element is reoriented more than once during the operation. Only shell elements adjacent to the lateral (side) faces are considered.

The command reorients the shell element normals on the same panel as the specified shell element. A *panel* is the geometry defined by a subset of shell elements bounded by free edges or T-junctions (anywhere three or more shell edges share common nodes).

Reorientation progresses within the selected set until either of the following conditions is true:

- The edge of the model is reached.
- More than two elements (whether selected or unselected) are adjacent to a lateral face.

In situations where unselected elements might undesirably cause case b to control, consider using **ENSYM**,0,,0,ALL instead of **ENORM**. It is recommended that reoriented elements be displayed and graphically reviewed.

You cannot use the **ENORM** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Real constant values are not reoriented and may be invalidated by an element reversal.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Shell Normals

---

ENSYM, *IINC*, --, *NINC*, *IEL1*, *IEL2*, *IEINC*

Generates elements by symmetry reflection.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

IINC
Increment to be added to element numbers in existing set.

--
Unused field.

NINC
Increment nodes in the given pattern by *NINC*.

IEL1, *IEL2*, *IEINC*
Reflect elements from pattern beginning with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and pattern is all selected elements [**ESEL**]. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *IEINC* are ignored).

Notes

This command is the same as the **ESYM** command except it allows explicitly assigning element numbers to the generated set (in terms of an increment *IINC*). Any existing elements already having these numbers will be re-defined.

The operation generates a new element by incrementing the nodes on the original element, and reversing and shifting the node connectivity pattern. For example, for a 4-node 2-D element, the nodes in positions I, J, K and L of the original element are placed in positions J, I, L and K of the reflected element.

Similar permutations occur for all other element types. For line elements, the nodes in positions I and J of the original element are placed in positions J and I of the reflected element. *In releases prior to ANSYS 5.5, no node pattern reversing and shifting occurred for line elements generated by* **ENSYM**. *To achieve the same results as you did in releases prior to ANSYS 5.5, use the* **ENGEN** *command instead.*

See the **ESYM** command for additional information about symmetry elements.

The **ENSYM** command also provides a convenient way to reverse shell element normals. If the *IINC* and *NINC* argument fields are left blank, the effect of the reflection is to reverse the direction of the outward normal of the specified elements. You cannot use the **ENSYM** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable. Also note that real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal. See Revising Your Model in the *ANSYS Modeling and Meshing Guide* for more information about controlling element normals.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Reverse Normals>of Shell Elems
Main Menu>Preprocessor>Modeling>Reflect>Elements>User Numbered

---

/EOF
Exits the file being read.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Notes

Causes an end-of-file exit when encountered on a switched file (see **/INPUT**, ***USE**, etc.). Commands are then read continuing from the file that contained the file switching command (or from the terminal if the switch was made from the terminal). Use the **/EXIT** command to terminate an ANSYS run. This command cannot be used in a do-loop or if-then-else construct.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

EORIENT, *Etype*, *Dir*, *TOLER*
Reorients solid element normals.

PREP7: Meshing
PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

Etype
Specifies which elements to orient.

LYSL
Specifies that only the layered solid elements will be oriented (SOLID46, SOLID95 with KEYOPT(1) = 1, and SOLID191). (Default.)

Dir

The axis and direction for orientation, or an element number. If *Dir* is set to a positive number (*n*), then all eligible elements are oriented as similarly as possible to element *n*.

NEGX
: The outward normal of face 1 is made as parallel as possible to the negative x-axis of the currently active element coordinate system.

POSX
: Same as NEGX, except using the positive x-axis.

NEGY
: The outward normal of face 1 is made as parallel as possible to the negative y-axis of the currently active element coordinate system.

POSY
: Same as NEGY, except using the positive y-axis.

NEGZ
: (Default) The outward normal of face 1 is made as parallel as possible to the negative z-axis of the currently active element coordinate system.

POSZ
: Same as NEGZ, except using the positive z-axis.

TOLER

The maximum angle (in degrees) between the outward normal face and the target axis. Default is 90.0. Lower *TOLER* values will reduce the number of faces that are considered as the basis of element reorientation.

Notes

EORIENT renumbers the elements to make the face 1 (nodes I-J-K-L, parallel to the layers) as parallel as possible to the XY plane of the element coordinate system (set with **ESYS**). It calculates the outward normal of each face and renumbers the nodes of the elements so the face whose normal is most parallel with and in the same general direction as the target axis becomes face 1.

The target axis, defined by *Dir*, is either the negative or positive indicated axis or the outward normal of face 1 of that element.

All SOLID46, SOLID95 with KEYOPT(1) = 1, and SOLID191 3-D layered solid elements in the selected set are considered for reorientation.

After reorienting elements, you should always display and graphically review results using the **/ESHAPE** command. When plotting models with many or symmetric layers, it may be useful to temporarily reduce the number of layers to two, with one layer being much thicker than the other.

You cannot use **EORIENT** to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Prisms and tetrahedrals are also supported, within the current limitations of the SOLID46, SOLID95, and SOLID191 elements (layers parallel to the 4-node face of the prism are not supported).

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Orient Normal

EPLOT

Produces an element display.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

Notes

Produces an element display of the selected elements. Only elements (or portions of elements) having nodes within the selected node set (and PowerGraphics off) [**NSEL**] are displayed. Adjacent or otherwise duplicated faces of 3-D solid elements will not be displayed. Duplicated faces normally occur in the interior of solid 3-D shapes. See the **DSYS** command for display coordinate system.

This command will display curvature in midside node elements when PowerGraphics is activated [**/GRAPHICS**,POWER] and **/EFACET**,2 or **/EFACET**,4 are enabled. (To display curvature, two facets per edge is recommended [**/EFACET**,2]). When you specify **/EFACET**,1, PowerGraphics does not display midside nodes. **/EFACET** has no effect on **EPLOT** for non-midside node elements.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking
Main Menu>Solution>Time Controls>Time Step Prediction
Utility Menu>Plot>Elements

EQSLV, *Lab*, *TOLER*, *MULT*

Specifies the type of equation solver.

SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> <> PP ED

Lab

Equation solver type:

FRONT

Frontal direct equation solver. In-memory only.

SPARSE

Sparse direct equation solver. Applicable to real symmetric and unsymmetric matrices. Available only for STATIC, HARMIC (full method only), TRANS (full method only), SUBSTR, and PSD spectrum analysis types [**ANTYPE**]. Can be used for nonlinear and linear analyses, especially nonlinear analysis where indefinite matrices are frequently encountered. Well suited for contact analysis where contact status alters the mesh topology. Other typical well-suited applications are: (a) models consisting of shell/beam or shell/beam and solid elements (b) models with a multi-branch structure, such as an automobile exhaust or a turbine fan. This is an alternative to iterative solvers since it combines both speed and robustness. Generally, it requires more memory than the frontal solver, but it is comparable in memory requirement to the PCG solver. When memory is limited, the solver works partly in and out-of-core memory without much increase in CPU time.

JCG

Jacobi Conjugate Gradient iterative equation solver, in-memory option. Available only for STATIC, MODAL (subspace option only), HARMIC (full method only), and TRANS (full method only) analysis types [**ANTYPE**]. Can be used for structural and multiphysics applications. Applicable for symmetric, unsymmetric, complex, definite, and indefinite matrices. Recommended for 3-D harmonic analyses in structural and multiphysics applications. Efficient for heat transfer, electromagnetics, piezoelectrics, and acoustic field problems.

JCGOUT

Jacobi Conjugate Gradient iterative equation solver, out-of-core memory option. Use this solver instead of JCG when you do not have sufficient real memory. Same as JCG except that it is valid only for symmetric, sparse, positive definite real matrices, and it operates from disk storage. Slower than JCG except for large problems that are run with small real memory (which causes the JCG solver to thrash). Not valid in a modal analysis. Recommended for heat transfer applications.

ICCG

Incomplete Cholesky Conjugate Gradient iterative equation solver. Available for STATIC, HARMIC (full method only), and TRANS (full method only) analysis types [**ANTYPE**]. Can be used for structural and multiphysics applications, and for symmetric, unsymmetric, complex, definite, and indefinite matrices. The ICCG solver requires more memory than the JCG solver, but is more robust than the JCG solver for ill-conditioned matrices.

PCG

Pre-conditioned Conjugate Gradient iterative equation solver (licensed from Computational Applications and Systems Integration, Inc.). Requires less disk file space than FRONT and is faster for large models (wavefront > about 1000). Much faster than the JCG solver. Useful for plates, shells, 3-D models, large 2-D models, p-method analyses, and other problems having symmetric, sparse, definite or indefinite matrices for nonlinear analysis. Requires twice as much memory as JCG (**/RUNST** can be used to determine the space needed). Available only for analysis types [**ANTYPE**] STATIC, TRANS (full method only), or MODAL (with subspace option only). The PCG solver can robustly solve equations with constraint equations (**CE**, **CEINTF**, **CPINTF**, and **CERIG**). This solver gives you the option of using either:

- the **MSAVE** command (used to obtain a considerable memory savings when applicable (see Notes below)), and/or
- the **PRECISION** command to specify single precision, thereby reducing the memory required by between 30% and 70%

PCGOUT

Pre-conditioned Conjugate Gradient iterative equation solver, out-of-memory option. Use this option instead of PCG when you do not have sufficient real memory. This solver is functionally the same as PCG except that the stiffness and preconditioned matrices are stored out-of-core memory. It is generally slower than the PCG solver due to the extra I/O needed during the solution process.

AMG

Algebraic Multigrid iterative equation solver. Available for STATIC analyses and TRANS (full method only) analyses. The AMG solver is applicable to symmetric matrices. This solver is not available for substructure analyses, either generation or use pass, or for models containing p-elements (which switch to the frontal solver). It is very efficient for single-field structural analyses (where the solution DOFs are combinations of UX, UY, UZ, ROTX, ROTY, and ROTZ). For applications such as single-field thermal analyses (where the solution DOF is TEMP), the AMG solver is less efficient. Recommended for ill-conditioned problems in which the PCG and ICCG solvers would have difficulty converging; applicable in both single- and multi-processor environments. In terms of CPU time when used in a single-processor environment, the AMG solver performs better than the PCG and ICCG solvers for ill-conditioned problems, and it delivers about

the same level of performance for ordinary problems. In a multiprocessor environment, the AMG solver scales better than the PCG and ICCG solvers on shared memory parallel machines. Also handles indefinite matrix problems for nonlinear analyses. The AMG solver is part of Parallel Performance for ANSYS, which is a separately-licensed product. For detailed information on multiprocessor solvers, see Improving ANSYS Performance and Parallel Performance for ANSYS in the *ANSYS Advanced Analysis Techniques Guide*. See Starting an ANSYS Session from Command Level in the *ANSYS Operations Guide* for information on how to specify the Parallel Performance for ANSYS add-on at ANSYS start-up.

ITER

Automatically chooses an iterative solver that is appropriate for the physics of the problem. The tolerance for the iterative solver is automatically selected based on the accuracy level you choose (see *TOLER*, below). This solver option is available only for electrostatic analyses, steady-state/transient thermal analyses and linear static/full transient structural analyses without superelements. If you issue the automatic iterative solver selection command, but the appropriate conditions for the selection are not met, then the program defaults to the Frontal solver. See Using the Automatic Iterative (Fast) Solver Option in the *ANSYS Basic Analysis Guide*, as well as Structural Static Analysis in the *ANSYS Structural Analysis Guide* or Steady-State Thermal Analysis in the *ANSYS Thermal Analysis Guide*, for correct usage of this option.

DDS

Distributed Domain Solver (DDS). Available for symmetric STATIC analyses and TRANS (full method only) analyses. Constraint equations (**CE**, **CEINTF**, and **CERIG**) and coupling (**CP**, **CPINTF**) are allowed. Not available for piezoelectric analyses.

The DDS solver works for deformable-to-deformable, surface-to-surface contact elements from 169 to 170, and other node-to-node contact elements. It does not work for rigid-to-deformable or node-to-surface contact elements in general. It also does not support elements with u-P formulation or Lagrangian multiplier as their theoretical bases.

The DDS solver works well with CE/CP and the CE/CP derived capabilities. However, if the CE/CP are specified at contact nodes, it may not work properly.

Note — When the solver does not work, the program will issue a FATAL error.

The solver decomposes a large problem into domains (internal substructures) and uses different processors or systems to solve each domain to reduce total solution time.

Use of DDS requires a license for the Parallel Performance for ANSYS advanced task (add-on). See Starting an ANSYS Session from Command Level in the *ANSYS Operations Guide* for information on how to specify the Parallel Performance for ANSYS add-on at ANSYS start-up. For more information on Parallel Performance for ANSYS, see Improving ANSYS Performance and Parallel Performance for ANSYS in the *ANSYS Advanced Analysis Techniques Guide*.

DPCG

Distributed Pre-conditioned Conjugate Gradient iterative equation solver. Based on an enhanced theory for the classic PCG solver, the DPCG solver targets distributed parallel processing. The DPCG solver preserves all of the merits of the classic PCG solver and can be run on either shared memory or distributed memory machines with superior scalability to the PCG solver. Compared to the DDS solver, the DPCG solver is more robust and uses less memory with similar scalability at low (less than 16) numbers of processors. The total sum of memory used by the DPCG (summed total over the network or all processors) is about 30% more than the classic PCG solver. In addition to the limitations of the classic PCG solver, the DPCG solver does not support subspace eigensolver (Powerdynamics), the **PRECISION** command, or p-elements.

Use of DPCG requires a license for the Parallel Performance for ANSYS advanced task (add-on). For more information on Parallel Performance for ANSYS in general, as well as more information on DPCG usage, see Improving ANSYS Performance and Parallel Performance for ANSYS in the *ANSYS Advanced Analysis Techniques Guide*.

DJCG

Distributed Jacobi Conjugate Gradient iterative equation solver. Based on an enhanced theory for the classic JCG solver, the DJCG solver targets distributed parallel processing. Scalability of this solver is superior to the JCG solver with little additional memory required. DJCG solver is available only for STATIC and TRANS (full method) analyses where the stiffness is symmetric. This solver does not support the fast thermal option (**THOPT**).

Use of DJCG requires a license for the Parallel Performance for ANSYS advanced task (add-on). For more information on Parallel Performance for ANSYS in general, as well as more information on DJCG usage, see Improving ANSYS Performance and Parallel Performance for ANSYS in the *ANSYS Advanced Analysis Techniques Guide*.

TOLER

Solver tolerance value (defaults to 1.0E-8) for static analyses with symmetric matrices. For unsymmetric static analyses or harmonic analyses (JCG and ICCG), as well as for the Domain solver (DDS), the default is 1.0E-6. The value 1.0E-5 may be acceptable in many situations.

Note — Using a tolerance less than 1.0E-6 with the Domain solver may lead to unconvergence. A tolerance of 1.0E-6 gives an accurate solution.

Used only with the Jacobi Conjugate Gradient, Incomplete Cholesky Conjugate Gradient, Pre-conditioned Conjugate Gradient, Algebraic Multigrid and Distributed Domain equation solvers. For all iterative solvers (except the PCG solver in the case of linear static analysis), iterations continue until the SRSS norm of the residual is less than *TOLER* times the norm of the applied load vector. (For the PCG solver in the static linear analysis case, 3 error norms are used. If one of the error norms is smaller than *TOLER*, and the SRSS norm of the residual is smaller than 1.0E-2, convergence is assumed to have been reached.) See the *ANSYS, Inc. Theory Reference* for details.

Note — When used with the Pre-conditioned Conjugate Gradient equation solver, *TOLER* can be modified between load steps (this is typically useful for nonlinear analysis).

If used with the ITER label, you can specify an integer value from 1 to 5, which indicates the accuracy level for convergence for the automatically-selected iterative solver. A value of 1 is the fastest level, and 5 is the most accurate level. ANSYS chooses the tolerance value for the automatic iterative solver based on the input accuracy level.

MULT

Multiplier (defaults to 2.0 when solution control is on; 1.0 when solution control is off) used to control the maximum number of iterations performed during convergence calculations. Used only with the Pre-conditioned Conjugate Gradient equation solver (PCG). The maximum number of iterations is equal to the multiplier (*MULT*) times the number of degrees of freedom (DOF). Iterations continue until either the maximum number of iterations or solution convergence has been reached. In general, the default value for *MULT* is adequate for reaching convergence. However, for ill-conditioned matrices (that is, models containing elements with high aspect ratios or material type discontinuities) the multiplier may be used to increase the maximum number of iterations used to achieve convergence. The recommended range for the multiplier is $1.0 \leq MULT \leq 3.0$. Normally, a value greater than 3.0 adds no further benefit toward convergence, and merely increases time requirements. If the solution does not converge with $1.0 \leq MULT \leq 3.0$, then convergence is highly unlikely, and further examination of the model is recommended.

Command Default

The sparse direct solver is the default solver for all analyses, except for electromagnetic analyses with CIRCU124 elements present, analyses that include both p-elements and constraint equations, spectrum analyses, and substructuring analyses (which each use the frontal direct solver by default).

Notes

The selection of a solver can affect the speed and accuracy of a solution. For a more detailed discussion of the merits of each solver, see Solution in the *ANSYS Basic Analysis Guide*.

If you are using the PCG solver to do a static analysis or modal analysis using the PowerDynamics method of a model containing SOLID92, SOLID95, SOLID186, and/or SOLID187 elements with linear material properties, you can save significant amounts of memory with the **MSAVE** command. **MSAVE**,ON will trigger an element-by-element approach for the parts of the model containing these element types with linear material properties (must be small strain when using SOLID186 and SOLID187 elements). (Other parts of the model will use global assembly for the stiffness matrix.) Using the element-by-element approach results in memory savings of up to 70% over the global assembly approach for the affected part of the model, although the solution time may be increased depending on the processor speed and computer hardware, as well as the chosen element options (for example, reduced 2 x 2 x 2 integration for SOLID95 is faster than default 14 point integration).

In a modal analysis, choosing the subspace mode extraction method along with the PCG or PCGOUT solver is the same as choosing the PowerDynamics mode extraction method. If you choose the PCGOUT solver, the solution will be much slower. The element-by element approach is also valid for the PowerDynamics method.

When doing a modal analysis with a large number of constraint equations, use the subspace iteration method with the frontal solver instead of the JCG solver, or use the Block Lanczos mode extraction method. (Block Lanczos is the default.)

If you use **MODOPT**,LANB or **BUCOPT**,LANB, the internal solver used is **EQSLV**,SPARSE.

Do not use the DDS solver for elements with high aspect ratios; doing so could result in convergence difficulties. The DDS solver is suitable for models with a combination of solids, trusses, beams, and shells.

If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Sol'n Options
Main Menu>Preprocessor>Loads>Fast Sol'n Optn
Main Menu>Solution>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Sol'n Controls>Sol'n Options
Main Menu>Solution>Fast Sol'n Optn

ERASE

Explicitly erases the current display.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Notes

Similar to a hardware screen erase key. Useful during an "immediate" display to erase the screen without a replot so that the display continues on a clean screen. This action is automatically included in commands such as **NPLOT** and **EPLOT**.

If the **/NOERASE** command is active, issuing the erase command will simply clear the display area. Subsequent replots will provide the display previously generated by the **/NOERASE** command.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Erase Options>Erase Screen

/ERASE

Specifies that the screen is to be erased before each display.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Command Default

Previous **/ERASE** or **/NOERASE** setting will be used. Initially defaults to **/ERASE** setting.

Notes

Erase occurs with the next display request, but before the display is actually started. **/NOERASE** can be used to suppress the automatic screen erase.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Erase Options>Erase Between Plots

EREAD, *Fname*, *Ext*, --

Reads elements from a file.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext

Filename extension (8 character maximum).

The extension defaults to ELEM if *Fname* is blank.

--

Unused field

Notes

This read operation is not necessary in a standard ANSYS run but is provided as a convenience to users wanting to read a coded element file, such as from another mesh generator or from a CAD/CAM program. Data should be formatted as produced with the **EWRITE** command. If issuing **EREAD** to acquire element information generated from ANSYS **EWRITE**, you must also issue **NREAD** before the **EREAD** command. The element types [**ET**] must be defined before the file is read so that the file may be read properly. Only elements that are specified with the **ERRANG** command are read from the file. Also, only elements that are fully attached to the nodes specified on the **NRRANG** command are read from the file. Elements are assigned numbers consecutively as read from the file, beginning with the current highest database element number plus one. The file is rewound before and after reading. Reading continues until the end of the file.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Read Elem File

---

EREFINE, *NE1*, *NE2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

Refines the mesh around specified elements.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NE1, *NE2*, *NINC*

Elements (*NE1* to *NE2* in increments of *NINC*) around which the mesh is to be refined. *NE2* defaults to *NE1*, and *NINC* defaults to 1. If *NE1* = ALL, *NE2* and *NINC* are ignored and all selected elements are used for refinement. If *NE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NE1* (*NE2* and *NINC* are ignored).

LEVEL

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

DEPTH
Depth of mesh refinement in terms of number of elements outward from the indicated elements, *NE1* to *NE2* (defaults to 0).

POST
Type of postprocessing to be done after element splitting, in order to improve element quality:

OFF
No postprocessing will be done.

SMOOTH
Smoothing will be done. Node locations may change.

CLEAN
Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

RETAIN
Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The ANSYS program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

ON
The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

OFF
The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

Notes

EREFINE performs local mesh refinement around the specified elements. By default, the surrounding elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

EREFINE refines all area elements and tetrahedral volume elements that are adjacent to the specified elements. Any volume elements that are adjacent to the specified elements, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes [**IC**], coupled nodes [**CP** family of commands], constraint equations [**CE** family of commands], or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. For additional restrictions on mesh refinement, see Revising Your Model in the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>All
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Elements

ERESX, *Key*

Specifies extrapolation of integration point results.

SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> <> PP ED

Key

Extrapolation key:

DEFA
: Extrapolate integration point results to the nodes for all elements except those with active plasticity, creep, or swelling nonlinearities (default).

YES
: Extrapolate integration point results to the nodes for all elements. Only linear solution data will be extrapolated for those elements with active plasticity, creep, or swelling nonlinearities.

NO
: Copy (do not extrapolate) integration point results to the nodes for all elements.

Command Default

Extrapolate integration point results to the nodes for all elements except those with active plasticity, creep, or swelling nonlinearities (default).

Notes

Specifies whether the solution results at the element integration points are extrapolated or copied to the nodes for element and nodal postprocessing. The structural stresses, elastic and thermal strains, field gradients, and fluxes are affected. Nonlinear data (plastic, creep, and swelling strains) are always copied to the nodes, never extrapolated. For shell elements, **ERESX** applies only to integration point results in the in-plane directions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Integration Pt
Main Menu>Solution>Load Step Opts>Output Ctrls>Integration Pt

---

ERNORM, *Key*

Controls error estimation calculations.

POST1: Controls

MP ME ST DY <> PR <> <> <> PP ED

Key

Control key:

ON
: Perform error estimation (default). This option is not valid for PowerGraphics.

OFF
: Do not perform error estimation.

Command Default

Error estimation calculations are performed by default unless PowerGraphics is enabled [**/GRAPHICS**,POWER].

Notes

Especially for thermal analyses, program speed increases if error estimation is suppressed. Therefore, it might be desirable to use error estimation only when needed. The value of the **ERNORM** key is not saved on **file.db**. Consequently, you need to reissue the **ERNORM** key after a **RESUME** if you wish to deactivate error estimation again.

Menu Paths

Main Menu>General Postproc>Options for Outp
Utility Menu>List>Results>Options

ERRANG, *EMIN*, *EMAX*, *EINC*

Specifies the element range to be read from a file.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

EMIN, *EMAX*, *EINC*

Elements with numbers from *EMIN* (defaults to 1) to *EMAX* (defaults to 99999999) in steps of *EINC* (defaults to 1) will be read.

Notes

Defines the element number range to be read [**EREAD**] from the element file. If a range is also implied from the **NRRANG** command, only those elements satisfying both ranges will be read.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Read Elem File

---

ESCHECK, *Sele*, *Levl*, *Defkey*

Perform element shape checking for a selected element set.

PREP7: Database
SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> FL PP ED

Sele

Specifies whether to select elements for checking:

(blank)

List all warnings/errors from element shape checking.

ESEL
Select the elements based on the *.Levl* criteria specified below.

Levl

WARN
Select elements producing warning and error messages.

ERR
Select only elements producing error messages (default).

Defkey
Specifies whether check should be performed on deformed element shapes. .

0 - Do not update node coordinates before performing shape checks (default).
1 - Update node coordinates using the current set of deformations in the database.

Notes

Shape checking will occur according to the current **SHPP** settings. Although **ESCHECK** is valid in all processors, *Defkey* uses the current results in the database. If no results are available a warning will be issued.

This command is also valid in PREP7, SOLUTION and POST1.

Menu Paths

Main Menu>General Postproc>Check Elem Shape>Sel Warning/Error Elements

---

ESEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS*
Selects a subset of elements.

DATABASE: Selecting
MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

ALL
Restore the full set.

NONE
Unselect the full set.

INVE
Invert the current set (selected becomes unselected and vice versa).

STAT
Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Item
Label identifying data, see ESEL - Valid Item and Component Labels. Some items also require a component label. If *Item* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to ELEM. If *Item* = STRA (straightened), elements are selected whose midside nodes do not conform to the curved line or non-flat area on which they should lie. (Such elements are sometimes formed during volume meshing [**VMESH**] in an attempt to avoid excessive element distortion.) You should graphically examine any such elements to evaluate their possible effect on solution accuracy.

Comp
Component of the item (if required). Valid component labels are shown in ESEL - Valid Item and Component Labels below.

VMIN
Minimum value of item range. Ranges are element numbers, attribute numbers, load values, or result values as appropriate for the item. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored).

VMAX
Maximum value of item range. *VMAX* defaults to *VMIN* for input values.

For result values, *VMAX* defaults to infinity if *VMIN* is positive, or to zero if *VMIN* is negative. If *VMIN* = *VMAX*, a tolerance of ±0.005 x *VMIN* is used, or ±1.0E-6 if *VMIN* = 0.0. If *VMAX* ≠ *VMIN*, a tolerance of ±1.0E-8x (*VMAX*-*VMIN*) is used.

VINC
Value increment within range. Used only with integer ranges (such as for element and attribute numbers). Defaults to 1. *VINC* cannot be negative.

KABS
Absolute value key:

0
Check sign of value during selection.

1
Use absolute value during selection (sign ignored).

Command Default

All elements are selected.

Notes

Selects elements based on values of a labeled item and component. For example, to select a new set of elements based on element numbers 1 through 7, use **ESEL**,S,ELEM,,1,7. The subset is used when the ALL label is entered (or implied) on other commands, such as **ELIST**,ALL. Only data identified by element number are selected. Selected data are internally flagged; no actual removal of data from the database occurs. Different element subsets cannot

be used for different load steps [**SOLVE**] in a **/SOLU** sequence. The subset used in the first load step will be used for all subsequent load steps regardless of subsequent **ESEL** specifications.

This command is valid in any processor.

Elements crossing the named path (see **PATH** command) will be selected. This option is only available in PREP7 and POST1. If no geometry data has been mapped to the path (i.e., via **PMAP** and **PDEF** commands), the path will assume the default mapping option (**PMAP**,UNIFORM) to map the geometry prior to selecting the elements. If an invalid path name is given, the **ESEL** command is ignored (status of selected elements is unchanged). If there are no elements crossing the path, the **ESEL** command will return zero elements selected.

ESEL - Valid Item and Component Labels

Valid Item and Component Labels ESEL, *Type,Item,Comp,VMIN,VMAX,VINC,KABS*

Item	Comp	Description
ELEM		Element number.
ADJ		Elements adjacent to element *VMIN* (*VMAX* and *VINC* fields are ignored). Only elements (of the same dimensionality) adjacent to lateral faces are considered. Progression continues until edge of model or until more than two elements are adjacent at a face.
TYPE		Element type number.
ENAME		Element name (or identifying number).
MAT		Material number.
REAL		Real constant number.
ESYS		Element coordinate system number.
PART		LS-DYNA part number (applicable only to ANSYS LS-DYNA)
LIVE		Active elements [**EALIVE**]. *VMIN* and *VMAX* fields are ignored, etc.
LAYER		Layer number (only composite elements with a nonzero thickness for the requested layer number are included). [**LAYER**]
SEC		Cross section ID number [**SECNUM**]
PINC		p-elements that are included in convergence checking.
PEXC		p-elements that are excluded from convergence checking.
STRA		Straightened. See description under Argument Descriptions' Item field above.
SFE	PRES	Element pressure.
	CONV	Element convection bulk temperature.
	HFLUX	Element heat flux.
	FSI	Element (acoustic) fluid-structure interaction flag.
	IMPD	Element (acoustic) impedance.
	SHLD	Surface shielding properties on high-frequency elements. (Select logic works only on the value of conductivity, not relative permeability.)
	MXWF	Element Maxwell force flag.
	CHRGS	Electric surface charge density.
	INF	Element infinite surface flag.
BFE	TEMP	Element temperature.
	FLUE	Element fluence.
	HGEN	Element heat generation rate.

Valid Item and Component Labels ESEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS*

Item	Comp	Description
	JS	Element current density, magnitude only.
	MVDI	Element magnetic virtual displacements flag.
	CHRGD	Electric charge density.
PATH	*Lab*	Selects all elements being crossed by the path with name *Lab* (see **PATH** command). If *Lab* = ALL then all elements related to all defined paths will be selected.
	Valid item and component labels for element result values are:	
ETAB	*Lab*	Any user-defined element table label (see **ETABLE** command).

Menu Paths

Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking
Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements
Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts
Utility Menu>Select>Entities

/ESHAPE, *SCALE*, *KEY*

Displays elements with shapes determined from the real constants or section definition.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

SCALE
Scaling factor:

0
Use simple display of line and area elements (default).

1
Use real constants or section definition to form a solid shape display of elements.

FAC
Multiply certain real constants, such as thickness, by *FAC* and use them to form a solid shape display of elements (*FAC* must be greater than 0.01).

KEY
Current shell thickness key:

0
Use current thickness in the displaced solid shape display of shell elements (valid for SHELL181, SHELL208, and SHELL209). Default.

1
Use initial thickness in the displaced solid shape display of shell elements.

Command Default

Use simple display of line and area elements (0).

Notes

Allows beams, shells, current sources, and certain special purpose elements to be displayed as solids with the shape determined from the real constants or section types. Elements are displayed with the **EPLOT** command. No checks for valid or complete input are made for the display.

SOLID65 elements are displayed with internal lines that represent rebar sizes and orientations (requires vector mode (**/DEVICE**) with a basic type of display (**/TYPE**,,BASIC)). The rebar with the largest volume ratio in each element plots as a red line, the next largest as green, and the smallest as blue.

COMBIN14, COMBIN39, and MASS21 are displayed with a graphics icon with the offset determined by the real constants and KEYOPT settings.

BEAM188 and BEAM189 are displayed as solids with the shape determined from the section definition commands, **SECTYPE** and **SECDATA**. The elements are displayed with internal lines representing the cross-section mesh. Contour plots are available for these elements in postprocessing for PowerGraphics only (**/GRAPHICS**,POWER). To view 3-D deformed shapes for BEAM188 and BEAM189, issue **OUTRES**,MISC or **OUTRES**,ALL for static or transient analyses. To view 3-D mode shapes for a modal or eigenvalue buckling analysis, you must expand the modes with element results calculation ON (*Elcalc* = YES for **MXPAND**).

When plotting stresses (**PLNSOL**,s,x) of BEAM4 and BEAM44, the thermal warping stresses are not included in the stress display. An example of element temperatures causing warping thermal stresses is -1, 1, -1, 1, -1, 1, -1, 1.

When **/ESHAPE** is active, displacement plots (via **PLNSOL**,U,X and **PLDISP**, for example) may disagree with listings generated via the **PRNSOL** command. To more realistically display a displaced model, ANSYS sometimes graphically rotates the model.

Also, when PowerGraphics is active, **/ESHAPE** may degrade the image if adjacent elements have overlapping material, such as shell elements, which are not coplanar.

/ESHAPE may also degrade the image if adjacent elements have different real constants when PowerGraphics is active. All polygons depicting the connectivity between the “thicker” and “thinner” elements along the shared element edges may not always be displayed.

In most cases, **/ESHAPE** renders a thickness representation of your shell, plane and layered elements more readily in PowerGraphics. This type of representation employs PowerGraphics to generate the enhanced representation, and will often provide no enhancement in Full Graphics. This is especially true for **/POST1** results displays, where **/ESHAPE** is not supported for most element types.

If you are using **/ESHAPE** and want to view solution results (**PLNSOL**, etc.) on layered elements (such as SHELL91, SHELL99, SOLID46, SHELL181, SOLID191, SHELL208, and SHELL209), set KEYOPT(8) = 1 for the layer elements so that the data for all layers is stored in the results file.

If PowerGraphics is active, you can plot the through-thickness temperatures of elements SHELL131 and SHELL132 regardless of the thermal DOFs in use by issuing the **PLNSOL**,TEMP command following the **/ESHAPE** command.

By default, **/ESHAPE** displays SOURC36, CIRCU124, and TRANS126 elements when PowerGraphics is activated (**/GRAPHICS**,POWER).

The **/ESHAPE**,1 and **/ESHAPE**,*FAC* commands are incompatible with the **/CYCEXPAND** command used in cyclic symmetry analyses.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

ESIZE, *SIZE*, *NDIV*

Specifies the default number of line divisions.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

SIZE
Default element edge length on surface boundaries (i.e., lines). Divisions are automatically calculated (rounded upward to next integer) from line lengths. If *SIZE* is zero (or blank), use *NDIV*.

NDIV
Default number of element divisions along region boundary lines. Not used if *SIZE* is input.

Notes

Specifies the default number of line divisions (elements) to be generated along the region boundary lines. The number of divisions may be defined directly or automatically calculated. Divisions defined directly for any line [**LESIZE**, **KESIZE**, etc.] are retained. For adjacent regions, the divisions assigned to the common line for one region are also used for the adjacent region. See the **MOPT** command for additional meshing options.

For free meshing operations, if smart element sizing is being used [**SMRTSIZE**] and **ESIZE**,*SIZE* has been specified, *SIZE* will be used as a starting element size, but will be overridden (i.e., a smaller size may be used) to accommodate curvature and small features.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Size
Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Adv Opts

---

ESLA, *Type*

Selects those elements associated with the selected areas.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of element select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

Notes

Selects area elements belonging to meshed [**AMESH**], selected [**ASEL**] areas.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

ESLL, *Type*

Selects those elements associated with the selected lines.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of element select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

Notes

Selects line elements belonging to meshed [**LMESH**], selected [**LSEL**] lines.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

ESLN, *Type*, *EKEY*, *NodeType*

Selects those elements attached to the selected nodes.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of element selected:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

EKEY
Node set key:

0
Select element if *any* of its nodes are in the selected nodal set (default).

1
Select element only if *all* of its nodes are in the selected nodal set.

NodeType
Label identifying type of nodes to consider when selecting:

ALL
Select elements considering all of their nodes (default).

ACTIVE
Select elements considering only their active nodes. An active node is a node that contributes DOFs to the model.

INACTIVE
Select elements considering only their inactive nodes (such as orientation or radiation nodes).

CORNER
Select elements considering only their corner nodes.

MID
Select elements considering only their midside nodes.

Notes

ESLN selects elements which have any (or all [**EKEY**]) *NodeType* nodes in the currently-selected set of nodes. Only elements having nodes in the currently-selected set can be selected.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements
Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts
Utility Menu>Select>Entities

ESLV, *Type*

Selects elements associated with the selected volumes.

DATABASE: Selecting
MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of element selected:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

Notes

Selects volume elements belonging to meshed [**VMESH**], selected [**VSEL**] volumes.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

ESOL, *NVAR*, *ELEM*, *NODE*, *Item*, *Comp*, *Name*

Specifies element data to be stored from the results file.

POST26: Set Up
MP ME ST DY <> PR EM <> FL PP ED

NVAR
Arbitrary reference number assigned to this variable (2 to *NV* [**NUMVAR**]). Overwrites any existing results for this variable.

ELEM
Element for which data are to be stored. If *ELEM* = P, graphical picking is enabled (valid only in the GUI).

NODE
Node number on this element for which data are to be stored. If blank, store the average element value (except for **FMAG** values, which are summed instead of averaged). If *NODE* = P, graphical picking is enabled (valid only in the GUI).

Item
Label identifying the item. General item labels are shown in ESOL - General Item and Component Labels below. Some items also require a component label.

Comp
Component of the item (if required). General component labels are shown in ESOL - General Item and Component Labels below. If *Comp* is a sequence number (*n*), the *NODE* field will be ignored.

Name
Thirty-two character name for identifying the item on the printout and displays. Defaults to a label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Notes

Valid item and component labels for element (except line element) results are listed in ESOL - General Item and Component Labels

Defines element results data to be stored from a results file [**FILE**]. Not all items are valid for all elements. See the input and output summary tables of the *ANSYS Elements Reference* of each element for the available items.

There are two methods of data access that may be used with the **ESOL** command. You can access some simply by using a generic label (Component Name method), while others require a label and number (Sequence Number method).

The Component Name method is used to access the General element data (that is, element data which is generally available to most element types or groups of element types). The General *Item* and *Comp* labels for the Component Name method are shown in ESOL - General Item and Component Labels below.

The Sequence Number method is required for data that is not averaged (such as pressures at nodes, temperatures at integration points, etc.), or data that is not easily described in a generic fashion (such as all derived data for structural line elements and contact elements, all derived data for thermal line elements, layer data for layered elements, etc.). A table illustrating the *Items* (such as LS, LEPEL, LEPTH, SMISC, NMISC, etc.) and corresponding sequence numbers for each element is shown in Chapter 4, "Element Library" in the *ANSYS Elements Reference*.

Element results are in the element coordinate system, except for layered elements where results are in the layer coordinate system. Results are obtainable for an element at a specified node. Further location specifications can be made for some elements with the **SHELL**, **LAYERP26**, and **FORCE** commands.

ESOL - General Item and Component Labels

General Item and Component Labels ESOL, *NVAR, ELEM, NODE, Item, Comp, Name*

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.

General Item and Component Labels ESOL, *NVAR,ELEM,NODE,Item,Comp,Name*

Item	Comp	Description
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1,2,3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
GKS	X	Gasket component stress (also gasket pressure).
GKD	X	Gasket component total closure.
GKDI	X	Gasket component total inelastic closure.
GKTH	X	Gasket component thermal closure.
CONT	STAT	Contact status. 3-closed and sticking, 2-closed and sliding, 1-open but near contact, 0-open and not near contact.
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress
"	STOT	Contact total stress (pressure plus friction)
"	SLIDE	Contact sliding distance
"	GAP	Contact gap distance
"	FLUX	Total heat flux at contact surface
"	CNOS	Total number of contact status changes during substep.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.

General Item and Component Labels ESOL, *NVAR, ELEM, NODE, Item, Comp, Name*

Item	Comp	Description
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component magnetic forces or vector sum.
F	X, Y, Z	Component structural force.
M	X, Y, Z	Component structural moment.
HEAT[1]		Heat flow.
FLOW		Fluid flow.
AMPS		Current flow.
FLUX		Magnetic flux.
VF	X, Y, Z	Component fluid force.
CSG	X, Y, Z	Component magnetic current segment.
SENE		"Stiffness" energy.
KENE		Kinetic energy.
JHEAT		Element Joule heat generation.
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).
JS	X, Y, Z, SUM	Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low frequency electric analyses. Components (X, Y, Z) and vector sum (SUM).
JT	X, Y, Z, SUM	Total measureable current density in low-frequency electromagnetic analyses. (Conduction current density in a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM).
MRE		Magnetics Reynolds number
VOLU		Volume of volume element.
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels HBOT, HE2, HE3, . . ., HTOP instead of HEAT.

Menu Paths

Main Menu>TimeHist Postpro>Define Variables
Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

ESORT, *Item*, *Lab*, *ORDER*, *KABS*, *NUMB*

Sorts the element table.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

Item
Label identifying the item:

ETAB
(currently the only *Item* available)

Lab
element table label:

Lab
Any user-defined label from the **ETABLE** command (input in the *Lab* field of the **ETABLE** command).

ORDER
Order of sort operation:

0
Sort into descending order.

1
Sort into ascending order.

KABS
Absolute value key:

0
Sort according to real value.

1
Sort according to absolute value.

NUMB
Number of elements (element table rows) to be sorted in ascending or descending order (*ORDER*) before sort is stopped (remainder will be in unsorted sequence) (defaults to all elements).

Command Default

Use ascending element number order.

Notes

The element table rows are sorted based on the column containing the *Lab* values. Use **EUSORT** to restore the original order. If **ESORT** is specified with PowerGraphics on [**/GRAPHICS**,POWER], then the nodal solution results listing [**PRNSOL**] will be the same as with the full graphics mode [**/GRAPHICS**,FULL].

Menu Paths

Main Menu>General Postproc>List Results>Sorted Listing>Sort Elems

ESSOLV, *Electit*, *Strutit*, *DIMN*, *MORPHOPT*, *Mcomp*, *Xcomp*, *ELECTOL*, *STRUTOL*, *MXLOOP*, *--*, *RUSEKY*, *RESTKY*, *EISCOMP*

Performs a coupled electrostatic-structural analysis.

SOLUTION: Analysis Options

MP <> <> <> <> <> <> <> <> <> ED

Electit

Title of the electrostatics physics file as assigned by the **PHYSICS** command.

Strutit

Title of the structural physics file as assigned by the **PHYSICS** command.

DIMN

Model dimensionality (a default is not allowed):

2

2-D model.

3

3-D model.

MORPHOPT

Morphing option:

<0

Do not perform any mesh morphing or remeshing.

0

Remesh the non-structural regions for each recursive loop only if mesh morphing fails (default).

1

Remesh the non-structural regions each recursive loop and bypass mesh morphing.

2

Perform mesh morphing only, do not remesh any non-structural regions.

Mcomp

Component name of the region to be morphed. For 2-D models, the component may be elements or areas. For 3-D models, the component may be elements or volumes. A component must be specified. You must enclose name-strings in single quotes in the **ESSOLV** command line.

Xcomp

Component name of entities excluded from morphing. In the 2-D case, it is the component name for the lines excluded from morphing. In the 3-D case, it is component name for the areas excluded from morphing. Defaults to exterior non-shared entities (see the **DAMORPH**, **DVMORPH**, and **DEMORPH** commands). You must enclose name-strings in single quotes in the **ESSOLV** command line.

ELECTOL

Electrostatic energy convergence tolerance. Defaults to .005 (.5%) of the value computed from the previous iteration. If less than zero, the convergence criteria based on electrostatics results is turned off.

STRUTOL

Structural maximum displacement convergence tolerance. Defaults to .005 (.5%) of the value computed from the previous iteration. If less than zero, the convergence criteria base on structural results is turned off.

MXLOOP
Maximum number of allowable solution recursive loops. A single pass through both an electrostatics and structural analysis constitutes one loop. Defaults to 100.

--
Unused field.

RUSEKY
Reuse flag option:

$\leq$1
Assumes initial run of **ESSOLV** using base geometry for the first electrostatics solution.

>1
Assumes **ESSOLV** run is a continuation of a previous **ESSOLV** run, whereby the morphed geometry is used for the initial electrostatic simulation.

RESTKY
Structural restart key.

0
Use static solution option for structural solution.

1
Use static restart solution option for structural solution.

EISCOMP
Element component name for elements containing initial stress data residing in file **jobname.ist**. The initial stress data must be defined prior to issuing **ESSOLV** (see **ISFILE** command).

Notes

ESSOLV invokes an ANSYS macro which automatically performs a coupled electrostatic-structural analysis.

The macro displays periodic updates of the convergence.

If non-structural regions are remeshed during the analysis, boundary conditions and loads applied to nodes and elements will be lost. Accordingly, it is better to assign boundary conditions and loads to the solid model.

Use RUSEKY>1 for solving multiple **ESSOLV** simulations for different excitation levels (i.e., for running a voltage sweep). Do not issue the **SAVE** command to save the database between **ESSOLV** calls.

For nonlinear structural solutions, the structural restart option (RESTKY = 1) may improve solution time by starting from the previous converged structural solution.

For solid elements, **ESSOLV** automatically detects the air-structure interface and applies a Maxwell surface flag on the electrostatic elements. This flag is used to initiate the transfer for forces from the electrostatic region to the structure. When using the **ESSOLV** command with structural shell elements (e.g., SHELL63, SHELL93), you must manually apply the Maxwell surface flag on all air elements surrounding the shells before writing the final electrostatic physics file. Use the **SFA** command to apply the Maxwell surface flag to the areas representing the shell elements. This will ensure that the air elements next to both sides of the shells receive the Maxwell surface flag.

Note — If lower-order structural solids or shells are used, set KEYOPT(7) = 1 for the electrostatic element types to ensure the correct transfer of forces.

Information on creating the initial stress file is documented in the Initial Stress Loading section in the *ANSYS Basic Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Physics>Coupled Solvers>Elec/struc
Main Menu>Solution>Physics>Coupled Solvers>Elec/struc

ESTIF, *KMULT*

Specifies the matrix multiplier for deactivated elements.

SOLUTION: Birth and Death

MP ME ST <> <> <> <> <> <> PP ED

KMULT
Stiffness matrix multiplier for deactivated elements (defaults to 1.0E-6).

Command Default

Use 1.0E-6 as the multiplier.

Notes

Specifies the stiffness matrix multiplier for elements deactivated with the **EKILL** command (birth and death).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Birth & Death>StiffnessMult
Main Menu>Solution>Load Step Opts>Other>Birth & Death>StiffnessMult

---

ESURF, *XNODE*, *Tlab*, *Shape*

Generates elements overlaid on the free faces of existing selected elements.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

XNODE
Used only when generating SURF151 or SURF152 elements with KEYOPT(5) = 1. *XNODE* is the single extra node number (ID) used with these surface elements. There is no default. *XNODE* must be specified if KEYOPT(5) = 1. If *XNODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A parameter or parametric expression can be substituted for *XNODE*.

Tlab
Used to generate target and contact elements with correct direction of normals. This label is used only with TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, and CONTA174 elements.

TOP
Generates target and contact elements over beam and shell elements with the normals the same as the underlying beam and shell elements (default).

BOTTOM
Generates target and contact elements over beam and shell elements with the normals opposite to the underlying beam and shell elements.

REVERSE
Reverses the direction of the normals on existing selected target and contact elements.

Shape
Used to specify the element shape for target element TARGE170.

(blank)
The target element takes the same shape as the external surface of the underlying element (default).

TRI
Generates several triangular facet target elements.

Notes

Generates elements of the currently active element type overlaid on the free faces of existing elements. For example, surface elements (SURF151, SURF152, SURF153, or SURF154) can be generated over solid elements (PLANE55, SOLID70, PLANE42, SOLID45, respectively). Element faces are determined from the selected node set [**NSEL**] and the load faces for that element type. The operation is similar to that used for generating element loads from selected nodes with the **SF**,ALL command, except that elements, instead of loads, are generated. All nodes on the face must be selected for the face to be used. For shell elements, only face one of the element is available. If nodes are shared by adjacent selected element faces, the faces are not free and no element will be generated. Elements created by **ESURF** will be oriented such that their surface load directions are consistent with those of the underlying elements. Generated elements, as well as their orientation, should be checked carefully. Generated elements use the existing nodes and the active MAT, TYPE, REAL, and ESYS attributes, except when *Tlab* = REVERSE. The reversed target and contact elements will have the same attributes as the original elements. If the underlying elements are solid elements, *Tlab* = TOP or BOTTOM has no effect.

By default, when the command generates a target element the shape will be the same as that of the underlying element. While not recommended, you can issue **ESURF**,,,TRI to generate several facet triangle elements.

ESURF is also used to generate 2-D or 3-D node-to-surface element CONTA175, based on the selected node components of the underlying solid elements. When used to generate CONTA175 elements, all **ESURF** arguments are ignored.

Regarding the GUI, if CONTA175 is the active element type, the path **Main Menu> Preprocessor> Modeling> Create> Elements> Node-to-Surf** will automatically use **ESURF** to generate elements.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Inf Acoustic
Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Node to Surf
Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Generl Surface>Extra Node
Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Generl Surface>No extra Node

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf to Surf

ESYM, --, *NINC*, *IEL1*, *IEL2*, *IEINC*

Generates elements from a pattern by a symmetry reflection.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

--
Unused field.

NINC
Increment nodes in the given pattern by *NINC*.

IEL1, *IEL2*, *IEINC*
Reflect elements from pattern beginning with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and pattern is all selected elements [**ESEL**]. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *IEINC* are ignored).

Notes

Generates additional elements from a given pattern (similar to **EGEN**) except with a "symmetry" reflection. The operation generates a new element by incrementing the nodes on the original element, and reversing and shifting the node connectivity pattern. For example, for a 4-node 2-D element, the nodes in positions I, J, K, and L of the original element are placed in positions J, I, L, and K of the reflected element.

Similar permutations occur for all other element types. For line elements, the nodes in positions I and J of the original element are placed in positions J and I of the reflected element. *In releases prior to ANSYS 5.5, no node pattern reversing and shifting occurred for line elements generated by* **ESYM**. *To achieve the same results with ANSYS 5.5 as you did in prior releases, use the* **EGEN** *command instead.*

It is recommended that symmetry elements be displayed and graphically reviewed.

If the nodes are also reflected (as with the **NSYM** command) this pattern is such that the orientation of the symmetry element remains similar to the original element (i.e., clockwise elements are generated from clockwise elements).

For a non-reflected node pattern, the reversed orientation has the effect of reversing the outward normal direction (clockwise elements are generated from counterclockwise elements).

Note — Since nodes may be defined anywhere in the model independently of this command, any orientation of the "symmetry" elements is possible. See also the **ENSYM** command for modifying existing elements.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Elements>Auto Numbered

ESYS, *KCN*

Sets the element coordinate system attribute pointer.

PREP7: Meshing
PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

KCN
Coordinate system number:

0
Use element coordinate system orientation as defined (either by default or by KEYOPT setting) for the element (default).

N
Use element coordinate system orientation parallel to local coordinate system *N* (where *N* must be greater than 10). For global system 0, 1, or 2, define a local system *N* parallel to appropriate system with the **LOCAL** or **CS** command (for example: **LOCAL**,11,1).

Command Default

Use element coordinate system orientation as defined (either by default or by KEYOPT setting) for the element (default).

Notes

Identifies an element coordinate system to be assigned to subsequently defined elements. Used only with area and volume elements. This number refers to the coordinate system reference number (*KCN*) defined using the **LOCAL** (or similar) command. Element coordinate system numbers may be displayed [**/PNUM**].

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs
Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

---

ET, *ITYPE*, *Ename*, *KOP1*, *KOP2*, *KOP3*, *KOP4*, *KOP5*, *KOP6*, *INOPR*

Defines a local element type from the element library.

PREP7: Element Type

MP ME ST DY <> PR EM EH FL PP ED

ITYPE
Arbitrary local element type number. Defaults to 1 + current maximum.

Ename
Element name (or number) as given in the element library in Chapter 4 of the *ANSYS Elements Reference*. The name consists of a category prefix and a unique number, such as BEAM3. The category prefix of the name (BEAM for the example) may be omitted but is displayed upon output for clarity. If *Ename* = 0, the element is defined as a null element.

KOP1, *KOP2*, *KOP3*, *KOP4*, *KOP5*, *KOP6*
KEYOPT values (1 through 6) for this element, as described in the *ANSYS Elements Reference*.

INOPR
If 1, suppress all element solution printout for this element type.

Notes

Elements types are selected from the element library (and are therefore established as local element types for the current model) by this command. Information derived from the element type is used for succeeding commands, so the **ET** command(s) should be entered early. The *ANSYS Elements Reference* describes the various elements which are available in the ANSYS library. For example, if a 2-D plane quadrilateral structural element type is desired, *Ename* should be PLANE42 or 42. A special option, *Ename* = 0, permits this element type to be ignored during solution without actually removing the element from the model. *Ename* may be set to zero only after the element type has been previously defined with a nonzero *Ename*. The preferred method of ignoring elements is to use the select commands (such as **ESEL**).

KOPn are element option keys. These keys (referred to as KEYOPT(*n*)) are used to turn on certain element options for this element. These options are listed under "KEYOPT" in the input table for each element type in the *ANSYS Elements Reference*. KEYOPT values include stiffness formulation options, printout controls, and various other element options. If KEYOPT(7) or greater is needed, input their values with the **KEYOPT** command.

The **ET** command only defines an element type local to your model (from the types in the element library). The **TYPE** or similar [**KATT**, **LATT**, **AATT**, or **VATT**] command must be used to point to the desired local element type before meshing.

To activate the ANSYS program's LS-DYNA explicit dynamic analysis capability, use the **ET** command or its GUI equivalent to choose an element that works only with LS-DYNA (such as SHELL163). Choosing LS-DYNA in the **Preferences** dialog box does *not* activate LS-DYNA; it simply makes items and options related to LS-DYNA accessible in the GUI.

Menu Paths

Main Menu>Preprocessor>Element Type>Add/Edit/Delete

ETABLE, *Lab*, *Item*, *Comp*

Fills a table of element values for further processing.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

Lab
Any unique user defined label for use in subsequent commands and output headings (maximum of eight characters and not a General predefined *Item* label). Defaults to an eight character label formed by concatenating the first four characters of the *Item* and *Comp* labels. If the same as a previous user label, this result item will be included under the same label. Up to 200 different labels may be defined. The following labels are predefined by ANSYS and cannot be used as user-defined labels: REFL, STAT, and ERAS. *Lab* = REFL refills all tables previously defined with the **ETABLE** commands (not the **CALC** module commands) according to the latest **ETABLE** specifications and is convenient for refilling tables after the load step [**SET**] has been changed. Remaining fields will be ignored if *Lab* is REFL. *Lab* = STAT displays stored table values. *Lab* = ERAS erases the entire table.

Item
Label identifying the item. General item labels are shown in the table below. Some items also require a component label. Character parameters may be used. *Item* = ERAS erases a *Lab* column.

Comp
Component of the item (if required). General component labels are shown in the table below. Character parameters may be used.

Notes

Defines a table of values per element (the element table) for use in further processing (see POST1 Element Table commands). The element table is organized as a "worksheet," with the rows representing all selected elements, and the columns consisting of result items which have been moved into the table (*Item*,*Comp*) with **ETABLE**. Each column of data is identified by a user-defined label (*Lab*) for listings and displays.

After entering the data into the element table, you are not limited to merely listing or displaying your data [**PLESOL**, **PRESOL**, etc.]. You may also perform many types of operations on your data, such as adding or multiplying columns [**SADD**, **SMULT**], defining allowable stresses for safety calculations [**SALLOW**], or multiplying one column by another [**SMULT**]. See the *ANSYS Basic Analysis Guide* for further details.

There are different types of results data that may be stored in the element table. For example, many items for an element are inherently single-valued (i.e., there is only one value per element). The single-valued items include: SERR, SDSG, TERR, TDSG, SENE, TENE, KENE, JHEAT, JS, VOLU, and CENT. All other items are multivalued (i.e., they vary over the element, such that there is a different value at each node). Since only one value is stored in the element table per element, an average value (based on the number of contributing nodes) is calculated for multivalued items. Exceptions to this averaging procedure are FMAG and all element force items, which represent the sum only of the contributing nodal values.

There are two methods of data access that may be used with the **ETABLE** command. The method you choose will depend upon the type of data that you wish to store. Some results can be accessed just with the use of a generic label (Component Name method), while others require a label and number (Sequence Number method).

The Component Name method is used to access the General element data (that is, element data which is generally available to most element types or groups of element types). All of the single-valued items and some of the more general multivalued items are accessible with the Component Name method. Various element results depend on the calculation method and the selected results location (**AVPRIN**, **RSYS**, **LAYER**, **SHELL**, and **ESEL**).

Although nodal data is readily available for listings and displays [**PRNSOL**, **PLNSOL**] without using the element table, you may also use the Component Name method to enter these results into the element table for further "worksheet" manipulation (see the *ANSYS Basic Analysis Guide* for more details). A listing of the General *Item* and *Comp* labels for the Component Name method is shown below.

The Sequence Number method allows you to view results for data that is not averaged (such as pressures at nodes, temperatures at integration points, etc.), or data that is not easily described in a generic fashion (such as all derived data for structural line elements and contact elements, all derived data for thermal line elements, layer data for layered elements, etc.). A table illustrating the *Items* (such as LS, LEPEL, LEPTH, SMISC, NMISC, SURF, etc.) and corresponding sequence numbers for each element is shown in the Output Data section of each element description found in the *ANSYS Elements Reference*.

Some element table data are reported in the results coordinate system. These include all component results (for example, UX, UY, etc.; SX, SY, etc.). The solution writes component results in the database and on the results file in the solution coordinate system. When you issue the **ETABLE** command, these results are then transformed into the results coordinate system [**RSYS**] before being stored in the element table. The default results coordinate

system is global Cartesian [**RSYS**,0]. All other data are retrieved from the database and stored in the element table with no coordinate transformation.

Use the **PRETAB**, **PLETAB**, or **ETABLE**,STAT commands to display the stored table values. Issue **ETABLE**,ERAS to erase the entire table. Issue **ETABLE**,*Lab*,ERAS to erase a *Lab* column.

When the GUI is on, if a **Delete** operation in a **Define Element Table Data** dialog box writes this command to a log file (**Jobname.LOG** or **Jobname.LGW**), you will observe that *Lab* is blank, *Item* = ERASE, and *Comp* is an integer number. In this case, the GUI has assigned a value of *Comp* that corresponds to the location of a chosen variable name in the dialog box's list. It is *not* intended that you type in such a location value for *Comp* in an ANSYS session. However, a file that contains a GUI-generated **ETABLE** command of this form can be used for batch input or for use with the **/INPUT** command.

ETABLE - General Item and Component Labels

General Item and Component Labels ETABLE, *Lab, Item, Comp*

Item	Comp	Description
Valid Item Labels for Degree of Freedom Results		
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.
TEMP[1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity.
A	X, Y, Z	X, Y, or Z magnetic vector potential.
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy.
ENDS		Turbulent energy dissipation.
SP0*n*		Mass fraction of species *n*, where *n* = 1 to 6. If a species is given a user-defined name [**MSSPEC**], use that name instead of SP0*n*.
Valid Item Labels for FLOTRAN Nodal Results		
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
ECON		Fluid effective conductivity.

General Item and Component Labels ETABLE, *Lab, Item, Comp*

Item	Comp	Description
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.
LMD*n*		Laminar mass diffusion coefficient for species *n*, where *n* = 1 to 6.
EMD*n*		Effective mass diffusion coefficient for species *n*, where *n* = 1 to 6.
Valid Item and Component Labels for Element Results		
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
EPSW		Swelling strain.
EPTO	X, Y, Z, XY, YZ, XZ	Component total mechanical strain (*excluding* thermal) (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total mechanical strain.
"	INT	Total mechanical strain intensity.
"	EQV	Total equivalent mechanical strain.
EPTT	X, Y, Z, XY, YZ, XZ	Component total strain *including* thermal (EPEL + EPTH + EPPL + EPCR).
"	1, 2, 3	Principal total strain.
"	INT	Total strain intensity.
"	EQV	Total equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable. VISCO106, VISCO107, and VISCO108 only.
"	PLWK	Plastic work/volume. VISCO106, VISCO107, and VISCO108 only.

General Item and Component Labels ETABLE, *Lab, Item, Comp*

Item	Comp	Description
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component magnetic forces or vector sum.
SERR		Structural error energy.
SDSG		Absolute value of maximum variation of any nodal stress component.
TERR		Thermal error energy.
TDSG		Absolute value of the maximum variation of any nodal thermal gradient component.
F	X, Y, Z	Component structural force. Sum of element nodal values.
M	X, Y, Z	Component structural moment. Sum of element nodal values.
HEAT		Heat flow. Sum of element nodal values.
FLOW		Fluid flow. Sum of element nodal values.
AMPS		Current flow. Sum of element nodal values.
FLUX		Magnetic flux. Sum of element nodal values.
VF	X, Y, Z	Component fluid force.
CSG	X, Y, Z	Component magnetic current segment.
SENE		"Stiffness" energy or thermal heat dissipation (applies to all elements where meaningful). Same as TENE.
AENE		Artificial energy of the element. This includes the sum of hourglass control energy and energy generated by in-plane drilling stiffness from shell elements (applies to all elements where meaningful). The energy is used for comparisons to SENE energy to predict the solution error due to artificial stiffness. See the *ANSYS, Inc. Theory Reference*.
TENE		Thermal heat dissipation or "stiffness" energy (applies to all elements where meaningful). Same as SENE.
KENE		Kinetic energy (applies to all elements where meaningful).
JHEAT		Element Joule heat generation.
JS	X, Y, Z, SUM	Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low frequency electric analyses. Components (X, Y, Z) and vector sum (SUM).
JT	X, Y, Z, SUM	Total measureable current density in low-frequency electromagnetic analyses. (Conduction current density in a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM).
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).

General Item and Component Labels ETABLE, *Lab, Item, Comp*

Item	Comp	Description
MRE		Magnetics Reynolds number
VOLU		Element volume. Based on unit thickness for 2-D plane elements (unless the thickness option is used) and on the full 360 degrees for 2-D axisymmetric elements.
CENT	X, Y, Z	Undeformed X, Y, or Z location (based on shape function) of the element centroid in the active coordinate system.
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
SMISC	*snum*	Element summable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each applicable element description in the *ANSYS Elements Reference*).
NMISC	*snum*	Element non-summable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each applicable element description found in the *ANSYS Elements Reference*).
SURF	*snum*	Element surface data value at sequence number *snum* (shown in Section 2.2.2.5: Surface Solution of the *ANSYS Elements Reference*).
CONT[2]	STAT	Contact status. 3-closed and sticking, 2-closed and sliding, 1-open but near contact, 0-open and not near contact.
"	PENE	Contact penetration (zero or positive).
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
"	GAP	Contact gap distance (0 or negative).
"	FLUX	Total heat flux at contact surface.
"	CNOS	Total number of contact status changes during substep.
TOPO		Densities used for topological optimization. This applies to the following types of elements: PLANE2, PLANE82, SOLID92, SHELL93, SOLID95.

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use labels TBOT, TE2, TE3, . . ., TTOP instead of TEMP.
2. For the CONT items for elements CONTA171 through CONTA175, the reported data is averaged across the element.

Menu Paths

Main Menu>General Postproc>Element Table>Define Table
Main Menu>General Postproc>Element Table>Erase Table

ETCHG, *Cnv*

Changes element types to their corresponding types.

PREP7: Element Type

MP ME ST DY <> PR EM EH FL PP ED

Cnv

Converts the element types to the corresponding type. Valid labels are:

ETI
Explicit to Implicit

ITE
Implicit to Explicit

TTE
Thermal to Explicit

TTS
Thermal to Structural

STT
Structural to Thermal

MTT
Magnetic to Thermal

FTS
Fluid to Structural

ETS
Electrical to Structural

Notes

Changes the currently defined element types to their corresponding types. Elements without a companion element (listed above) are not switched and should be switched with the **ET** command to an appropriate element type or to a null element. The KEYOPT values for the switched element types are reset to zero or to their default values. You must check these values to see if they are still meaningful. Additionally, if *Cnv* = ETI, ITE, or TTE, all real constants are set to zero.

If *Cnv* = ITE, you will need to choose a material model that corresponds to your previously-defined material properties. If working interactively, you will be prompted to do so.

If no conversion type is specified, the command returns to pre-ANSYS 5.4 behavior (that is, the command behaves as if *Cnv* were set to TTS, STT, MTT, FTS, and ETS, but the KEYOPT values are not reset).

Element types LINK8 and LINK10 lack a third node; however, their corresponding companion explicit element types, LINK160 and LINK167, require a third (orientation) node. If you are using element types LINK8 or LINK10, you must first convert the element type using this command and then manually define the third node of LINK160 or LINK167 elements using **EMODIF**.

Also, if you are converting BEAM4 to BEAM161, you may need to manually define the third node of BEAM161 elements as well. However, BEAM4 allows you to define a third, optional node. If you have defined this third node on BEAM4, then the conversion to BEAM161 will be completed automatically when you issue this command. If you did not define the third node on BEAM4, then you must manually define it on BEAM161 using **EMODIF**.

Element Pairs

ETI -- Explicit to Implicit

160 > 8	163 > 181	165 > 14	167 > 10
161 > 4	164 > 185	166 > 21	

ITE -- Implicit to Explicit

4 > 161	10 > 167	21 > 166	185 > 164
8 > 160	14 > 165	181 > 163	

TTE --Thermal to Explicit

55 > 162	57 > 163	70 > 164

TTS -- Thermal to Structural

32 > 1	67 > 42	75 > 25	131 > 181
33 > 8	68 > 8	77 > 82	132 > 91/93
35 > 2	69 > 45	78 > 83	151 > 153
55 > 42	70 > 45	87 > 92	152 > 154
57 > 41	71 > 21	90 > 95	157 > 63

STT -- Structural to Thermal

1 > 32	62 > 70	88 > 77	154 > 152
2 > 35	63 > 131	89 > 90	158 > 87
8 > 33	64 > 70	91 > 132	180 > 33
21 > 71	65 > 70	92 > 87	181 > 131
25 > 75	73 > 70	93 > 132	182 > 55
41 > 57	74 > 77	95 > 90	183 > 77
42 > 55	82 > 77	106 > 55	185 > 70
45 > 70	83 > 78	107 > 70	186 > 90
56 > 55	84 > 55/77	108 > 77	187 > 87
58 > 70	86 > 70	153 >151	

MTT -- Magnetic to Thermal

53 > 77	96 > 70	117 > 90	120 > 90
62 > 70	97 > 70	119 > 87	

FTS -- Fluid to Structural

141 > 42	142 > 45

ETS -- Electrical to Structural

67 > 42	121 > 42/82	123 > 92	128 > 45/95
68 > 8	122 > 45/95	127 > 92	157 > 63
69 > 45			

Menu Paths

Main Menu>Preprocessor>Element Type>Switch Elem Type

ETCONTROL, *Eltech*, *Eldegene*

Control the element technologies used in element formulation (for applicable elements).

PREP7: Element Type

MP ME ST DY <> PR EM EH FL PP ED

Eltech

Element technology control (SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, SHELL209).

SUGGESTION

ANSYS makes a suggestion for the best element technology before solving. If necessary, mixed u-P (KEYOPT(6)) will also be included and reset (default).

SET

ANSYS informs you of the best settings and automatically resets any applicable KEYOPT settings; this will override any manual settings you provided previously.

OFF

Turns automatic selection of element technology off. No suggestions will be issued, and no automatic resetting is done

Eldegene

Element degenerated shape control (PLANE182, PLANE183, SOLID185, SOLID186).

ON

If element shapes are degenerated, the degenerated shape function is employed and enhanced strain and B-bar formulations are turned off (default).

OFF

If element shapes are degenerated, regular shape functions are still used, and the specified element technologies (e.g., enhanced strain, B-bar, uniform reduced integration) are still used.

Notes

The command default is: ETCONTROL,Suggestion,ON. This command is valid for: SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209. See Section 2.17: Automatic Selection of Element Technologies in the *ANSYS Elements Reference* for more details.

Menu Paths

Main Menu>Preprocessor>Element Type>Elem Tech Control

ETDELE, *ITYP1*, *ITYP2*, *INC*

Deletes element types.

PREP7: Element Type

MP ME ST DY <> PR EM <> FL PP ED

ITYP1, *ITYP2*, *INC*

Deletes element types from *ITYP1* to *ITYP2* (defaults to *ITYP1*) in steps of *INC* (defaults to 1). If *ITYP1* = ALL, *ITYP2* and *INC* are ignored and all element types are deleted. Element types are defined with the **ET** command.

Menu Paths

Main Menu>Preprocessor>Element Type>Add/Edit/Delete

ETLIST, *ITYP1*, *ITYP2*, *INC*

Lists currently defined element types.

PREP7: Element Type

MP ME ST DY <> PR EM <> FL PP ED

ITYP1, *ITYP2*, *INC*

Lists element types from *ITYP1* to *ITYP2* (defaults to *ITYP1*) in steps of *INC* (defaults to 1). If *ITYP1* = ALL (default), *ITYP2* and *INC* are ignored and all element types are listed.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Properties>Element Types

ETYPE

Specifies "Element types" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Element Types

EUSORT

Restores original order of the element table.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

Notes

Changing the selected element set [**ESEL**] also restores the original element order.

Menu Paths

Main Menu>General Postproc>List Results>Sorted Listing>Unsort Elems

---

*EVAL, *ETYPE*, *EVPARM*, *CONST*, *XMIN*, *XMAX*, *XVAL*, *ECALC*

Evaluates hyperelastic constants.

PREP7: Materials

MP ME ST DY <> <> <> <> <> PP ED

ETYPE

The type of hyperelastic constant to evaluate (only one is currently available):

1

Mooney-Rivlin

EVPARM

The type of hyperelastic equations (mode of deformation) to be used in the evaluation:

1

Uniaxial (tension or compression) equations.

2

Equibiaxial (tension or compression) equations.

3

Shear (Planar Tension or Compression) equations.

CONST

Name of the array parameter containing the determined constants that are to be evaluated. This array must have been previously filled with hyperelastic constant values using the ***MOONEY** command.

XMIN

Minimum engineering strain value for which determined hyperelastic constants will be evaluated. *XMIN* must have a value greater than -1.

XMAX

Maximum engineering strain value for which determined hyperelastic constants will be evaluated.

XVAL

Name of the array parameter containing the engineering strain values at which the hyperelastic constants will be evaluated. This array must have been previously defined [***DIM**] to have dimensions *P*x1, where *P* is the number of calculated data points (equally spaced over the range *XMIN* to *XMAX*) that will be used to

define the curve. *P* should typically be given a fairly large value in order to generate a smooth curve that accurately reflects the true curve shape.

ECALC
Name of the array parameter in which evaluated engineering stress values are placed. This array must have been previously defined [***DIM**] to have dimensions *P*x1, where *P* is defined above.

Notes

Evaluates the hyperelastic constants over a specified range of strain values after the ***MOONEY** command is used. Constants can be used to evaluate engineering stress data outside the range of the input (experimental) stress-strain data. The evaluated stress-strain curves can be displayed by using the ***VPLOT** command to graph *ECALC* (evaluated stress array parameter) versus *XVAL* (strain array parameter).

You can also plot evaluated stress-strain curves if you have only Mooney-Rivlin constants (and no laboratory test data). To do an ***EVAL** in this case, you need to dimension [***DIM**] and fill the *CONST* array with the Mooney-Rivlin constants. You can fill this array fairly easily, given that the *CONST* array is either 1 x 2, 1 x 5, or 1 x 9 at most.

Menu Paths

This command cannot be accessed from a menu.

EWRITE, *Fname*, *Ext*, --, *KAPPND*, *Format*

Writes elements to a file.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to ELEM if *Fname* is blank.

--
Unused field

KAPPND
Append key:

0
Rewind file before the write operation.

1
Append data to the end of the existing file.

Format
Format key:

SHORT
I6 format (the default).

LONG
I8 format.

Notes

Writes the selected elements to a file. The write operation is not necessary in a standard ANSYS run but is provided as convenience to users wanting a coded element file. If issuing **EWRITE** from ANSYS to be used in ANSYS, you must also issue **NWRITE** to store nodal information for later use. Only elements having all of their nodes defined (and selected) are written. Data are written in a coded format. The data description of each record is: I, J, K, L, M, N, O, P, MAT, TYPE, REAL, SECNUM, ESYS, IEL, where MAT, TYPE, REAL, and ESYS are attribute numbers, SECNUM is the beam section number, and IEL is the element number.

The format is (14I6) if *Format* is set to SHORT and (14I8) if the *Format* is set to LONG, with one element description per record for elements having eight nodes of less. For elements having more than eight nodes, nodes nine and above are written on a second record with the same format.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Write Elem File

---

/EXIT, *Slab*, *Fname*, *Ext*, --

Stops the run and returns control to the system.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Slab
Mode for saving the database:

MODEL
Save the model data (solid model, finite element model, loadings, etc.) only (default).

SOLU
Save the model data and the solution data (nodal and element results).

ALL
Save the model data, solution data and post data (element tables, path results, etc.)

NOSAVE
Do not save any data on **File.DB** (an existing DB file will not be overwritten).

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name, defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to DB if *Fname* is blank.

--
Unused field

Notes

The current database information may be written on **File.DB** or a named file. If **File.DB** already exists, a backup file (**File.DBB**) will also be written whenever a new **File.DB** is written.

This command is valid in any processor. Issuing this command at any point will exit the program.

Menu Paths

Utility Menu>File>Exit

EXP, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*, *FACTB*

Forms the exponential of a variable.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA
Reference number of the variable to be operated on.

--, --
Unused fields.

Name
Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --
Unused fields.

FACTA
Scaling factor applied to variable *IA* (defaults to 1.0).

FACTB
Scaling factor (positive or negative) applied to the operation (defaults to 1.0).

Notes

Forms the exponential of a variable according to the operation:

$$IR = FACTB * \mathrm{EXP}(FACTA \times IA)$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Exponentiate

EXPAND, *Nrepeat*, MODAL, *HIndex*, *Icsys*, *SctAng*, --, *Phase*
Displays the results of a modal cyclic symmetry analysis.

POST1: Special Purpose

MP ME ST DY <> PR <> <> <> PP ED

Nrepeat
Number of sector repetitions for expansion. The default is 0 (no expansion).

MODAL
Specifies that the expansion is for a modal cyclic symmetry analysis.

HIndex
The harmonic index ID for the results to expand.

Icsys
The coordinate system number used in the modal cyclic symmetry solution. The default is the global cylindrical coordinate system (specified via the **CSYS** command where *KCN* = 1).

SctAng
The sector angle in degrees, equal to 360 divided by the number of cyclic sectors.

--
This field is reserved for future use.

Phase
The phase angle in degrees to use for the expansion. The default is 0. Typically, the value is the peak displacement (or stress/strain) phase angle obtained via the **CYCPHASE** command.

Notes

Issue this command to display the results of a modal cyclic symmetry analysis.

When you issue the **EXPAND**,*Nrepeat* command, subsequent **SET** commands read data from the results file and expand them to *Nrepeat* sectors. As long as no entities have been modified, this expansion can be negated (that is, reverted to single sector) by issuing **EXPAND** with no arguments. If you modify entities and wish to return to the partial model, use the Session Editor (see Restoring Database Contents in the *ANSYS Operations Guide*).

EXPAND displays the results and allows you to print them, as if for a full model. The harmonic index (automatically retrieved from the results file) appears in the legend column.

When plotting kinetic energy (KENE) or strain energy (SENE), the **EXPAND** command works with brick or tet models only.

EXPAND is a specification command valid only in POST1. It is significantly different from the **/EXPAND** command in several respects, (although you can use either command to display the results of a modal cyclic symmetry analysis):

- **EXPAND** has none of the limitations of the **/EXPAND** command.
- **EXPAND** changes the database by modifying the geometry, the nodal displacements, and element stresses as they are read from the results file, whereas the **/EXPAND** command does not change the database.
- You can print results displayed via **EXPAND**.

Caution: The **EXPAND** command creates new nodes and elements; therefore, saving (or issuing the **/EXIT**, ALL command) after issuing the **EXPAND** command can result in large databases.

Menu Paths

Main Menu>General Postproc>Cyc Expansion

/EXPAND, *Nrepeat1*, *Type1*, *Method1*, *DX1*, *DY1*, *DZ1*, *Nrepeat2*, *Type2*, *Method2*, *DX2*, *DY2*, *DZ2*, *Nrepeat3*, *Type3*, *Method3*, *DX3*, *DY3*, *DZ3*

Allows the creation of a larger graphic display than represented by the actual finite element analysis model.

POST1: Special Purpose

MP ME ST DY <> PR <> <> <> PP ED

Nrepeat1, *Nrepeat2*, *Nrepeat3*
The number of repetitions required for the element pattern. The default is 0 (no expansion).

Type1, *Type2*, *Type3*
The type of expansion requested.

RECT
Causes a Cartesian transformation of DX, DY, and DZ for each pattern (default).

POLAR
Causes a polar transformation of DR, D-Theta and DZ for each pattern.

AXIS
Causes 2-D axisymmetric expansion (that is, rotates a 2-D model created in the X-Y plane about the Y axis to create a 3-D model).

LRECT
Causes a Cartesian transformation of DX, DY, and DZ for each pattern about the current local coordinate system (specified via the **CSYS** command).

LPOLAR
Causes a polar transformation of DR, D-Theta, and DZ for each pattern about the local coordinate system (specified via the **CSYS** command).

Method1, *Method2*, *Method3*
The method by which the pattern is repeated.

FULL
Causes a normal repeat of the pattern (default).

HALF
Uses a symmetry transformation for alternate repeats (to produce an image of a complete circular gear from the image of half a tooth, for example).

DX1, *DY1*, *DZ1*, *DX2*, *DY2*, *DZ2*, *DX3*, *DY3*, *DZ3*
The Cartesian or polar increments between the repeated patterns. Also determines the reflection plane. Reflection is about the plane defined by the normal vector (DX, DY, DZ). If you want no translation, specify a small nonzero value. For a half-image expansion, the increment DX, DY, or DZ is doubled so that POLAR,HALF, ,45 produces full images on 90° centers, and RECT,HALF, ,1 produces full images on 2-meter centers.

Notes

You can use the **/EXPAND** command to perform up to three symmetry expansions at once (that is, X, Y, and Z which is equal to going from a 1/8 model to a full model). Polar expansions allow you to expand a wheel section into a half wheel, then into the half section, and then into the whole.

The command displays elements/results when you issue the **EPLOT** command or postprocessing commands.

The command works on all element and result displays, except as noted below. As the graphic display is created, the elements (and results) are repeated as many times as necessary, expanding the geometry and, if necessary, the displacements and stresses.

Derived results are not supported.

The **/EXPAND** command has the following limitations:

- It does not support solid model entities.
- POLAR, FULL or HALF operations are meaningful only in global cylindrical systems and are unaffected by the **RSYS** or **DSYS** commands. Cartesian symmetry or unsymmetric operations also occur about the global Cartesian system.
- It does not average nodal results across sector boundaries, even for averaged plots (such as those obtained via the **PLNSOL** command).

The **/EXPAND** command differs significantly from the **EXPAND** command in several respects:

- The uses of **/EXPAND** are of a more general nature, whereas the **EXPAND** command is intended *primarily* to expand modal cyclic symmetry results.
- **/EXPAND** does not change the database as does the **EXPAND** command.
- You cannot print results displayed via **/EXPAND**.

Menu Paths

Utility Menu>PlotCtrls>Style>Symmetry Expansion>
Utility Menu>PlotCtrls>Style>Symmetry Expansion>Expansion by values
Utility Menu>PlotCtrls>Style>Symmetry Expansion>Modal Cyclic Symmetry
Utility Menu>PlotCtrls>Style>Symmetry Expansion>Periodic/Cyclic Symmetry Expansion

---

EXPASS, *Key*

Specifies an expansion pass of an analysis.

SOLUTION: Analysis Options

MP ME ST <> <> PR <> <> <> PP ED

Key

Expansion pass key:

OFF

No expansion pass will be performed (default).

ON
An expansion pass will be performed.

Notes

Specifies that an expansion pass of a modal, substructure, buckling, transient, or harmonic analysis is to be performed.

Note — This separate solution pass requires an explicit **FINISH** to preceding analysis and reentry into SOLUTION.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>ExpansionPass
Main Menu>Solution>Analysis Type>ExpansionPass

EXPSOL, *LSTEP*, *SBSTEP*, *TIMFRQ*, *Elcalc*

Specifies the solution to be expanded for reduced analyses.

SOLUTION: Load Step Options

MP ME ST <> <> PR <> <> <> PP ED

LSTEP, *SBSTEP*
Expand the solution identified as load step *LSTEP* and substep *SBSTEP*.

TIMFRQ
As an alternative to *LSTEP* and *SBSTEP*, expand the solution at, or nearest to, the time value *TIMFRQ* (for **ANTYPE**,TRANS or **ANTYPE**,SUBSTR) or frequency value *TIMFRQ* (for **ANTYPE**,HARMIC). *LSTEP* and *SBSTEP* should be blank.

Elcalc
Element calculation key:

YES
Calculate element results, nodal loads, and reaction loads.

NO
Do not calculate these items.

Notes

Specifies the solution to be expanded from analyses that use reduced or mode superposition methods (**ANTYPE**,HARMIC, TRANS, or SUBSTR). Use the **NUMEXP** command to expand a group of solutions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Load Step
Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq

Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Load Step
Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq

EXTOPT, *Lab*, *Val1*, *Val2*, *Val3*

Controls options relating to the generation of volume elements from area elements.

PREP7: Volumes

MP ME ST DY <> PR EM <> FL PP ED

Lab

Label identifying the control option. The meanings of *Val1*, *Val2*, and *Val3* will vary depending on *Lab*.

ON

Sets carryover of the material attributes, real constant attributes, and element coordinate system attributes of the pattern area elements to the generated volume elements. Sets the pattern area mesh to clear when volume generations are done. *Val1*, *Val2*, and *Val3* are ignored.

OFF

Removes all settings associated with this command. *Val1*, *Val2*, and *Val3* are ignored.

STAT

Shows all settings associated with this command. *Val1*, *Val2*, and *Val3* are ignored.

ATTR

Sets carryover of particular pattern area attributes (materials, real constants, and element coordinate systems) of the pattern area elements to the generated volume elements. (See 2.) *Val1* can be:

0

Sets volume elements to use current **MAT** command settings.

1

Sets volume elements to use material attributes of the pattern area elements.

Val2 can be:

0

Sets volume elements to use current **REAL** command settings.

1

Sets volume elements to use real constant attributes of the pattern area elements.

Val3 can be:

0

Sets volume elements to use current **ESYS** command settings.

1

Sets volume elements to use element coordinate system attributes of the pattern area elements.

ESIZE

Val1 sets the number of element divisions in the direction of volume generation or volume sweep. For **VDRAG** and **VSWEEP**, *Val1* is overridden by the **LESIZE** command *NDIV* setting. *Val2* sets the spacing ratio (bias) in the direction of volume generation or volume sweep. If positive, *Val2* is the nominal ratio of last division size to first division size (if > 1.0, sizes increase, if < 1.0, sizes decrease). If negative, *Val2*

is the nominal ratio of center division(s) size to end divisions size. Ratio defaults to 1.0 (uniform spacing). *Val3* is ignored.

ACLEAR
Sets clearing of pattern area mesh. (See 3.) *Val1* can be:

0
Sets pattern area to remain meshed when volume generation is done.

1
Sets pattern area mesh to clear when volume generation is done. *Val2* and *Val3* are ignored.

VSWE
Indicates that volume sweeping options will be set using *Val1* and *Val2*. Settings specified with **EXTOPT**,VSWE will be used the next time the **VSWEEP** command is invoked. If *Lab* = VSWE, *Val1* becomes a label. *Val1* can be:

AUTO
Indicates whether you will be prompted for the source and target used by **VSWEEP** or if VSWE should automatically determine the source and target. If *Val1* = AUTO, *Val2* is ON by default. VSWE will automatically determine the source and target for **VSWEEP**. You will be allowed to pick more than one volume for sweeping. When *Val2* = OFF, the user will be prompted for the source and target for **VSWEEP**. You will only be allowed to pick one volume for sweeping.

TETS
Indicates whether **VSWEEP** will tet mesh non-sweepable volumes or leave them unmeshed. If *Val1* = TETS, *Val2* is OFF by default. Non-sweepable volumes will be left unmeshed. When *Val2* = ON, the non-sweepable volumes will be tet meshed if the assigned element type supports tet shaped elements.

Val3 is ignored for *Lab* = VSWE.

Val1, *Val2*, *Val3*
Additional input values as described under each option for *Lab*.

Notes

1. **EXTOPT** controls options relating to the generation of volume elements from pattern area elements using the **VEXT**, **VROTAT**, **VOFFST**, **VDRAG**, and **VSWEEP** commands. (When using **VSWEEP**, the pattern area is referred to as the source area.)
2. Enables carryover of the attributes of the pattern area elements to the generated volume elements when you are using **VEXT**, **VROTAT**, **VOFFST**, or **VDRAG**. (When using **VSWEEP**, since the volume already exists, use the **VATT** command to assign attributes before sweeping.)
3. When you are using **VEXT**, **VROTAT**, **VOFFST**, or **VDRAG**, enables clearing of the pattern area mesh when volume generations are done. (When you are using **VSWEEP**, if selected, the area meshes on the pattern (source), target, and/or side areas clear when volume sweeping is done.)
4. Neither **EXTOPT**,VSWE,AUTO nor **EXTOPT**,VSWE,TETS will be affected by **EXTOPT**,ON or **EXTOPT**, OFF.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volume Sweep>Sweep Opts
Main Menu>Preprocessor>Modeling>Operate>Extrude>Elem Ext Opts

EXTREM, *NVAR1*, *NVAR2*, *NINC*
Lists the extreme values for variables.

POST26: Listing

MP ME ST DY <> PR EM <> FL PP ED

NVAR1, *NVAR2*, *NINC*
List extremes for variables *NVAR1* through *NVAR2* in steps of *NINC*. Variable range defaults to its maximum. *NINC* defaults to 1.

Notes

Lists the extreme values (and the corresponding times) for stored and calculated variables. Extremes for stored variables are automatically listed as they are stored. Only the real part of a complex number is used. Extreme values may also be assigned to parameters [***GET**].

Menu Paths

Main Menu>TimeHist Postpro>List Extremes

F Commands

F, *NODE*, *Lab*, *VALUE*, *VALUE2*, *NEND*, *NINC*
Specifies force loads at nodes.

SOLUTION: FE Forces

MP ME ST <> <> PR EM <> <> PP ED

NODE
Node at which force is to be specified. If ALL, *NEND* and *NINC* are ignored and forces are applied to all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab
Valid force label. Structural labels: FX, FY, or FZ (forces); MX, MY, or MZ (moments). Thermal labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow). Fluid labels: FLOW (fluid flow). Electric labels: AMPS (current flow), CHRG (electric charge). Magnetic labels: FLUX (magnetic flux); CSGX, CSGY, or CSGZ (magnetic current segments). FLOTRAN labels: FX, FY, or FZ (forces).

VALUE
Force value or table name reference for specifying tabular boundary conditions. To specify a table, enclose the table name in percent signs (%), e.g., **F**, *NODE*,HEAT,%*tabname*%). Use the ***DIM** command to define a table.

VALUE2
Second force value (if any). If the analysis type and the force allow a complex input, *VALUE* (above) is the real component and *VALUE2* is the imaginary component.

NEND, *NINC*
Specifies the same values of force at the nodes ranging from *NODE* to *NEND* (defaults to *NODE*), in steps of *NINC* (defaults to 1).

Notes

The available force loads per node correspond to the degrees of freedom listed under "Degrees of Freedom" in the input table for each element type in the *ANSYS Elements Reference*. If both a force and a constrained degree of freedom [**D**] are specified at the same node, the constraint takes precedence. Forces are defined in the nodal coordinate system. The positive directions of structural forces and moments are along and about the positive nodal axis directions. The node and the degree of freedom label corresponding to the force must be selected [**NSEL**, **DOFSEL**].

Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following labels: Fluid (FLOW), Electric (AMPS), Structural force (FX, FY, FZ, MX, MY, MZ), FLOTRAN (FX, FY, FZ), and Thermal (HEAT, HBOT, HE2, HE3, . . ., HTOP). All labels are valid only in static (**ANTYPE**,STATIC) and full transient (**ANTYPE**,TRANS) analyses.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharge>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCurrent>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>ImprCurr>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Node Components
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>NodePSD>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flow>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharge>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCurrent>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>ImprCurr>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Nodes
Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Node Components
Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Nodes
Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Nodes
Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>NodePSD>On Nodes
Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Nodes
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flow>On Nodes

---

/FACET, *Lab*

Specifies the facet representation used to form solid model displays.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

Lab

Valid labels:

FINE
: Use finer tessellation to increase the number of facets for the display. Provides the best representation (but decreases speed of operation).

NORML
: Use the basic number of facets for the display (default).

COAR
: Use a limited number of facets for the display. This option will increase the speed of the operations, but may produce poor representations for some imported models.

WIRE
: Display model with a wireframe representation (fast, but surfaces will not be shown).

Command Default

Basic number of facets.

Notes

Specifies the facet (or polygon) representation used to form solid model displays. Used only with the **APLOT**, **ASUM**, **VPLOT**, and **VSUM** commands.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Solid Model Facets

FATIGUE

Specifies "Fatigue data status" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Fatigue Calcs

FC, *MAT*, *Lab1*, *Lab2*, *DATA1-6*

Provides failure criteria information and activates a data table to input temperature-dependent stress and strain limits.

PREP7: Materials
POST1: Failure Criteria

MP ME ST DY <> PR EM EH <> PP ED

MAT
Material reference number. You can define failure criteria for up to ten different materials.

Lab1
Type of data.

TEMP
Temperatures. Each of the materials you define can have a different set of temperatures to define the failure criteria.

EPEL
Strains.

S
Stresses.

Lab2
Specific criteria. Not used if *Lab1* = TEMP.

XTEN
Allowable tensile stress or strain in the x-direction. (Must be positive.)

XCMP
Allowable compressive stress or strain in the x-direction. (Defaults to negative of XTEN.)

YTEN
Allowable tensile stress or strain in the y-direction. (Must be positive.)

YCMP
Allowable compressive stress or strain in the y-direction. (Defaults to negative of YTEN.)

ZTEN
Allowable tensile stress or strain in the z-direction. (Must be positive.)

ZCMP
Allowable compressive stress or strain in the z-direction. (Defaults to negative of ZTEN.)

XY
Allowable XY stress or shear strain. (Must be positive.)

YZ
Allowable YZ stress or shear strain. (Must be positive.)

XZ
Allowable XZ stress or shear strain. (Must be positive.)

XYCP
XY coupling coefficient (Used only if *Lab1* = S). Defaults to -1.0.

YZCP
YZ coupling coefficient (Used only if *Lab1* = S). Defaults to -1.0.

XZCP
XZ coupling coefficient (Used only if *Lab1* = S). Defaults to -1.0.

DATA1 - 6
Description of *DATA1* through *DATA6*.

T1, T2, T3, T4, T5, T6
Temperature at which limit data is input. Used only when *Lab1* = TEMP.

V1, V2, V3, V4, V5, V6
Value of limit stress or strain at temperature T1 through T6. Used only when *Lab1* = S or EPEL.

Notes

The data table can be input in either PREP7 or POST1. This table is used only in POST1. When you postprocess failure criteria results defined using the **FC** command (**PLESOL**, **PRNSOL**, **PRRSOL**, etc.), the active coordinate system must be the coordinate system of the material being analyzed. You do this using **RSYS**, SOLU. For layered applications, you also use the **LAYER** command. See the specific element documentation in the *ANSYS Elements Reference* for information on defining your coordinate system for layers.

Some plotting and printing functions will not support Failure Criteria for your PowerGraphics displays. This could result in minor changes to other data when Failure Criteria are applied. See the appropriate plot or print command documentation for more information .

Menu Paths

Main Menu>General Postproc>Failure Criteria>Add/Edit
Main Menu>General Postproc>Failure Criteria>Temp Variation
Main Menu>Preprocessor>Material Props>Failure Criteria>Add/Edit
Main Menu>Preprocessor>Material Props>Failure Criteria>Temp Variation

---

FCCHECK

Checks both the strain and stress input criteria for all materials.

PREP7: Materials
POST1: Failure Criteria

MP ME ST DY <> PR EM <> FL PP ED

Notes

Issue the **FCCHECK** command to check the completeness of the input during the input phase.

Menu Paths

Main Menu>General Postproc>Failure Criteria>Criteria Check
Main Menu>Preprocessor>Material Props>Failure Criteria>Criteria Check

---

FCDELE, *MAT*

Deletes previously defined failure criterion data for the given material.

PREP7: Materials
POST1: Failure Criteria

MP ME ST DY <> PR EM <> <> PP ED

MAT
Material number. Deletes all **FC** command input for this material.

Notes

This command is also valid in POST1.

Menu Paths

Main Menu>General Postproc>Failure Criteria>Delete
Main Menu>Preprocessor>Material Props>Failure Criteria>Delete

---

FCLIST, *MAT*, *--*,*TEMP*

To list what the failure criteria is that you have input.

PREP7: Materials
POST1: Failure Criteria

MP ME ST DY <> PR EM EH <> PP ED

MAT
Material number (defaults to ALL for all materials).

--
Unused field

TEMP
Temperature to be evaluated at (defaults to TUNIF).

Notes

This command allows you to see what you have already input for failure criteria using the FC commands.

Menu Paths

Main Menu>General Postproc>Failure Criteria>List
Main Menu>Preprocessor>Material Props>Failure Criteria>List

FCUM, *Oper*, *RFACT*, *IFACT*

Specifies that force loads are to be accumulated.

SOLUTION: FE Forces

MP ME ST <> <> PR EM <> <> PP ED

Oper
Accumulation key:

REPL
Subsequent values replace the previous values (default).

ADD
Subsequent values are added to the previous values.

IGNO
Subsequent values are ignored.

RFACT
Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

IFACT
Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

Command Default

Replace previous values.

Notes

Allows repeated force load (force, heat flow, etc.) values to be replaced, added, or ignored. Operations apply to the selected nodes [NSEL]. and the force labels corresponding to the selected force labels [**DOFSEL**]. The operations occur when the next force specifications are defined. For example, issuing the command **F**,1,FX,250 after a previous **F**,1,FX,200 causes the current value of the force on node 1 in the x-direction to be 450 with the add operation, 250 with the replace operation, or 200 with the ignore operation. Scale factors are also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a force of 700. Scale factors are applied even if no previous values exist. Issue **FCUM**,STAT to show the current label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

Note — FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

FCUM does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Forces
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Forces

---

FDELE, *NODE*, *Lab*, *NEND*, *NINC*

Deletes force loads on nodes.

SOLUTION: FE Forces

MP ME ST DY <> PR EM <> <> PP ED

NODE
Node for which force is to be deleted. If ALL, *NEND* and *NINC* are ignored and forces are deleted on all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab
Valid force label. If ALL, use all appropriate labels. Structural labels: FX, FY, or FZ (forces); MX, MY, or MZ (moments). Thermal labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow). Fluid labels: FLOW (fluid flow). Electric labels: AMPS (current flow), CHRG (electric charge). Magnetic labels: FLUX (magnetic flux); CSGX, CSGY, or CSGZ (magnetic current segments). FLOTRAN labels: FX, FY, or FZ (forces).

NEND, *NINC*
Delete forces from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1).

Notes

The node and the degree of freedom label corresponding to the force must be selected [**NSEL**, **DOFSEL**].

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Forces>On All Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharge>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCurrent>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>ImprCurr>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Flow>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Node Components
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>NodePSD>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flow>On Nodes
Main Menu>Solution>Define Loads>Delete>All Load Data>All Forces>On All Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharge>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCurrent>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>ImprCurr>On Nodes
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Flow>On Nodes
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Nodes
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Node Components
Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>NodePSD>On Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Nodes
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flow>On Nodes

---

/FDELE, *Ident*, *Stat*

Deletes a binary file after it is used.

SESSION: Files

MP ME ST <> <> PR EM <> <> PP ED

Ident

ANSYS file name identifier. Valid identifiers are: EMAT, ESAV, FULL, SUB, MODE, TRI, DSUB, USUB, OSAV, and SELD. See the *ANSYS Basic Analysis Guide* for file descriptions.

Stat

Keep or delete key:

KEEP
: Keep this file.

DELE
: Delete (or do not write, if not necessary) this file.

Command Default

Keep all files.

Notes

Deletes as soon as possible (or prevents writing) a binary file created by the ANSYS program to save space.

Warning: Deleting files that are necessary for the next substep, load step, or analysis will prevent continuation of the run.

This command is valid only at the Begin Level.

Menu Paths

Utility Menu>File>ANSYS File Options

FE, *NEV*, *CYCLE*, *FACT*, *Title*

Defines a set of fatigue event parameters.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

NEV
Reference number for this event (within *MXEV*).

CYCLE
Number of required cycles (defaults to 1). If -1, erase all parameters and fatigue stresses for this event.

FACT
Scale factor to be applied to all loadings in this event (defaults to 1.0).

Title
User defined identification title for this event (up to 20 characters).

Command Default

Event assigned one cycle, unity scale factor, and no title.

Notes

Repeat **FE** command to define additional sets of event parameters (*MXEV* limit), to redefine event parameters, or to delete event stress conditions.

The set of fatigue event parameters is associated with all loadings and all locations. See the **FTSIZE** command for the maximum set of events (*MXEV*) allowed.

Menu Paths

Main Menu>General Postproc>Fatigue>Assign Events
Main Menu>General Postproc>Fatigue>Erase Event Data

FEBODY

Specifies "Body loads on elements" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Body Loads

FECONS

Specifies "Constraints on nodes" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>DOF Constraints

FEFOR

Specifies "Forces on nodes" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Forces

FELIST, *NEV1*, *NEV2*, *NINC*

Lists the fatigue event parameters.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

NEV1, *NEV2*, *NINC*

List event parameters from *NEV1* (defaults to 1) to *NEV2* (defaults to *NEV1*) in steps of *NINC* (defaults to 1). If *NEV1* = ALL, *NEV2* and *NINC* are ignored and all events are listed. Fatigue event parameters are defined with the **FE** command.

Menu Paths

Main Menu>General Postproc>Fatigue>List Event Data

FESURF

Specifies "Surface loads on elements" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Surface Loads

FILE, *Fname*, *Ext*, --

Specifies the data file where results are to be found.

POST1: Set Up
POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

If *Fname* is blank, the extension defaults to RST (for structural, fluid, or coupled-field analyses), to RTH (for thermal or electrical analyses), to RMG (for magnetic analyses), or to RFL (for FLOTRAN analyses). For postprocessing reduced structural analyses in POST26, use the RDSP extension for displacements from transient dynamic analyses or the RFRQ extension from harmonic response analyses. For postprocessing contact results corresponding to the initial contact state in POST1, use the RCN extension.

--
Unused field

Command Default

Use the result file with the **Jobname** as *Fname* and with the extension corresponding to the analysis type.

Notes

Specifies the ANSYS data file where the results are to be found for postprocessing.

Menu Paths

Main Menu>General Postproc>Data & File Opts
Main Menu>TimeHist Postpro>Settings>File
Utility Menu>File>List>Binary Files
Utility Menu>List>Files>Binary Files

FILEAUX2, *Fname*, *Ident*, --
Specifies the binary file to be dumped.

AUX2: Binary Files

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to the current **Jobname** if *Ident* is specified.

Ident
ANSYS filename identifier. See the *ANSYS Basic Analysis Guide* for file descriptions and identifiers. If not an ANSYS identifier, *Ident* will be used as the filename extension.

--
Unused field

Notes

Specifies the binary file to be dumped with the **DUMP** command.

Menu Paths

Utility Menu>File>List>Binary Files
Utility Menu>List>Files>Binary Files

FILEAUX3, *Fname*, *Ext*, --

Specifies the results file to be edited.

AUX3: Results Files

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to the current **Jobname** if *Ext* is specified.

Ext
Filename extension (8 character maximum).

--
Unused field

Notes

Specifies the results file to be edited.

Menu Paths

This command cannot be accessed from a menu.

FILEDISP, *Fname*, *Ext*, --

Specifies the file containing the graphics data.

DISPLAY: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Ext
Filename extension (8 character maximum).

--
Unused field

Notes

Specifies the input file containing the graphics data (defaults to **File.GRPH**).

Menu Paths

It is part of the DISPLAY program.

FILL, *NODE1*, *NODE2*, *NFILL*, *NSTRT*, *NINC*, *ITIME*, *INC*, *SPACE*

Generates a line of nodes between two existing nodes.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

NODE1, *NODE2*
Beginning and ending nodes for fill-in. *NODE1* defaults to next to last node specified, *NODE2* defaults to last node specified. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NFILL
Fill *NFILL* nodes between *NODE1* and *NODE2* (defaults to |*NODE2*-*NODE1*|-1). *NFILL* must be positive.

NSTRT
Node number assigned to first filled-in node (defaults to *NODE1* + *NINC*).

NINC
Add this increment to each of the remaining filled-in node numbers (may be positive or negative). Defaults to the integer result of (*NODE2*-*NODE1*)/(*NFILL* + 1), i.e., linear interpolation. If the default evaluates to zero, or if zero is input, *NINC* is set to 1.

ITIME, *INC*
Do fill-in operation a total of *ITIMEs*, incrementing *NODE1*, *NODE2* and *NSTRT* by *INC* each time after the first. *ITIME* and *INC* both default to 1.

SPACE
Spacing ratio. Ratio of last division size to first division size. If > 1.0, divisions increase. If < 1.0, divisions decrease. Ratio defaults to 1.0 (uniform spacing).

Notes

Generates a line of nodes (in the active coordinate system) between two existing nodes. The two nodes may have been defined in any coordinate system. Nodal locations and rotation angles are determined by interpolation. Any number of nodes may be filled-in and any node number sequence may be assigned. See the **CSCIR** command when filling across the 180° singularity line in a non-Cartesian system.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>Fill between Nds

---

FILLDATA, *IR*, *LSTRT*, *LSTOP*, *LINC*, *VALUE*, *DVAL*

Fills a variable by a ramp function.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Define data table as variable *IR* (2 to *NV* [**NUMVAR**]).

LSTRT
Start at location *LSTRT* (defaults to 1).

LSTOP
Stop at location *LSTOP* (defaults to maximum location as determined from data previously stored.

LINC
Fill every *LINC* location between *LSTRT* and *LSTOP* (defaults to 1).

VALUE
Value assigned to location *LSTRT*.

DVAL
Increment value of previous filled location by *DVAL* and assign sum to next location to be filled (may be positive or negative.)

Notes

Locations may be filled continuously or at regular intervals (LINC). Previously defined data at a location will be overwritten.

Menu Paths

Main Menu>TimeHist Postpro>Table Operations>Fill Data

/FILNAME, *Fname*, *Key*

Changes the Jobname for the analysis.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Fname
Name (32 characters maximum) to be used as the **Jobname**. Defaults to the initial **Jobname** as specified on the ANSYS execution command, or to **File** if none specified.

Key
Specify whether to use the existing log and error files or start new files.

0, OFF
Use existing log and error files.

1, ON
Start new log and error files (old files are closed but **not** deleted).

Notes

All subsequently created files will be named with this **Jobname** if *Key* = 0. Use *Key* = 1 to start new log and error files. The previous **Jobname** is typically defined on the ANSYS program execution line (see the *ANSYS Operations Guide*). This command is useful when different groups of files created throughout the run are to have different names. For example, the command may be used before each substructure pass to avoid overwriting files or having to rename each file individually.

This command is valid only at the Begin level.

Menu Paths

Utility Menu>File>Change Jobname

FINISH

Exits normally from a processor.

SESSION: Processor Entry
DISPLAY: Action

MP ME ST DY <> PR EM <> FL PP ED

Notes

Exits any of the ANSYS processors or the DISPLAY program. For the ANSYS processors, data will remain intact in the database but the database is not automatically written to a file (use the **SAVE** command to write the database to a file). See also the **/QUIT** command for an alternate processor exit command. If exiting POST1, POST26, or OPT, see additional notes below.

POST1: Data in the database will remain intact, including the POST1 element table data, the path table data, the fatigue table data, and the load case pointers.

POST26: Data in the database will remain intact, except that POST26 variables are erased and specification commands (such as **FILE**, **PRTIME**, **NPRINT**, etc.) are reset. Use the **/QUIT** command to exit the processor and bypass these exceptions.

OPT: Current optimization data are written to **File.OPT** for possible resume later [**OPRESU**]. See also the **OPSAVE** command to write optimization data.

This command is valid in any processor. This command is not valid at the Begin level.

Menu Paths

Main Menu>DesignXplorer VT>Solution>Solve
Main Menu>Finish

FITEM, *NFIELD*, *ITEM*, *ITEMY*, *ITEMZ*

Identifies items chosen by a picking operation (GUI).

DATABASE: Picking

MP ME ST DY <> PR EM <> FL PP ED

NFIELD

Field number on the command which uses the picking data. (Count the command name as a field, so that a 2 indicates the first command argument, 3 the second command argument, etc.) The corresponding field on the command will have a P51X label.

ITEM

Entity number of the entity picked. Negative entity numbers are used to indicate a range of entities. If the item picked is a coordinate location, then this field represents the X-coordinate. See also the **FLST** command.

ITEMY, *ITEMZ*

Y and Z coordinates of a picked coordinate location. *ITEM* represents the X coordinate. See also the **FLST** command.

Notes

This is a command generated by the GUI and will appear in the log file (**Jobname.LOG**) if graphical picking is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

On the log file, a set of **FITEM** commands is preceded by one **FLST** command which defines the picking specifications for that pick operation. The data listed in the **FITEM** commands are used by the first subsequent command containing a P51X label in one of its fields.

Caution: For a given entity type, a list containing an *ITEM* that is larger than the maximum defined entity, could deplete the system memory and produce unpredictable results.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

FJ, *ELEM*, *LABEL*, *VALUE*

Specify forces or moments on the components of the relative motion of a joint element.

SOLUTION: FE Forces

MP ME ST DY <> PR EM EH <> PP ED

ELEM

Element number or ALL to specify all joint elements.

LABEL

Valid labels:

FX

Force in local x direction.

FY
Force in local y direction.

FZ
Force in local z direction.

MX
Moment about local x axis.

MY
Moment about local y axis.

MZ
Moment about local z axis.

VALUE
Value of the label.

Notes

Valid for MPC184 (joint options in KEYOPT(1)).

See **FJDELE** for information on deleting forces and moments.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Joint Elems
Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Joint Elems

---

FJDELE, *ELEM*, *LAB*

Deletes forces (or moments) on the components of the relative motion of a joint element.

SOLUTION: FE Forces

MP ME ST DY <> PR EM EH <> PP ED

ELEM
Element number, or ALL. (leaving this blank defaults to ALL)

LABEL
Valid labels are:

FX
Force in local x direction.

FY
Force in local y direction.

FZ
Force in local z direction.

MX
Moment about local x axis.

MY
Moment about local y axis.

MZ
Moment about local z axis.

ALL, or (blank)
Delete all valid forces or moments.

Notes

Valid for MPC184 (joint options in KEYOPT(1)).

See **FJ** for information on specifying forces (or moments).

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Forces>On Joint Elems
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Joint Elems
Main Menu>Solution>Define Loads>Delete>All Load Data>All Forces>On Joint Elems
Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Joint Elems

---

FJLIST, *Elem*

Lists forces and moments applied on joint elements.

SOLUTION: FE Forces

MP ME ST <> <> PR EM <> <> PP ED

Elem
Element number or ALL (or blank). Lists joint element forces and moments on the specified element(s).

Notes

Valid for MPC184. See **FJ** for information on specifying forces and moments.

Menu Paths

Utility Menu>List>Loads>Joint Element Forces>On Picked Element

---

FK, *KPOI*, *Lab*, *VALUE*, *VALUE2*

Defines force loads at keypoints.

SOLUTION: Solid Forces

MP ME ST <> <> PR EM <> <> PP ED

KPOI
Keypoint at which force is to be specified. If ALL, apply to all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Lab
Valid force label. Structural labels: FX, FY, or FZ (forces); MX, MY, or MZ (moments). Thermal labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow). Fluid labels: FLOW (fluid flow). Electric labels: AMPS (current flow),

CHRG (electric charge). Magnetic labels: FLUX (magnetic flux); CSGX, CSGY, or CSGZ (magnetic current segments).

VALUE
Force value or table name reference for specifying tabular boundary conditions. To specify a table, enclose the table name in percent signs (%), e.g., **FK**, *KPOI*, HEAT,%*tabname*%). Use the ***DIM** command to define a table.

VALUE2
Second force value (if any). If the analysis type and the force allow a complex input, *VALUE* (above) is the real component and *VALUE2* is the imaginary component.

Notes

Forces may be transferred from keypoints to nodes with the **FTRAN** or **SBCTRAN** commands. See the **F** command for a description of force loads.

Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following labels: Fluid (FLOW), Electric (AMPS), Structural force (FX, FY, FZ, MX, MY, MZ), and Thermal (HEAT, HBOT, HE2, HE3, . . ., HTOP).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharge>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCurrent>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>ImprCurr>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>NodePSD>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flow>On Keypoints
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharge>On Keypoints
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCurrent>On Keypoints
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>ImprCurr>On Keypoints
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Keypoints
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Keypoints
Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Keypoints
Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Keypoints
Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>NodePSD>On Keypoints
Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Keypoints
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flow>On Keypoints

FKDELE, *KPOI*, *Lab*

Deletes force loads at a keypoint.

SOLUTION: Solid Forces

MP ME ST <> <> PR EM <> <> PP ED

KPOI

Keypoint at which force is to be deleted. If ALL, delete forces at all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Lab

Valid force label. If ALL, use all appropriate labels. See the **FDELE** command for labels.

Notes

Deletes force loads (and all corresponding finite element loads) at a keypoint. See the **FDELE** command for details.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Forces>On All KPs
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharge>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCurrent>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>ImprCurr>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Flow>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>NodePSD>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flow>On Keypoints
Main Menu>Solution>Define Loads>Delete>All Load Data>All Forces>On All KPs
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharge>On Keypoints
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCurrent>On Keypoints
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>ImprCurr>On Keypoints
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Flow>On Keypoints
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Keypoints
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>NodePSD>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Keypoints
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flow>On Keypoints

FKLIST, *KPOI*, *Lab*

Lists the forces at keypoints.

SOLUTION: Solid Forces

MP ME ST <> <> PR EM <> <> PP ED

KPOI

List forces at this keypoint. If ALL (default), list for all selected keypoints [**KSEL**]. If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Lab

Force label to be listed (defaults to ALL). See the **DOFSEL** command for labels.

Notes

Listing applies to the selected keypoints [**KSEL**] and the selected force labels [**DOFSEL**].

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Forces>On All Keypoints
Utility Menu>List>Loads>Forces>On Picked KPs

FL, *NLOC*, *NODE*, *SCFX*, *SCFY*, *SCFZ*, *Title*

Defines a set of fatigue location parameters.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

NLOC

Reference number for this location (within *MXLOC*). When defining a new location, defaults to lowest unused location. If the specified *NODE* is already associated with a location, *NLOC* defaults to that existing location.

NODE

Node number corresponding to this location (must be unique). Used only to associate a node with a new location or to find an existing location (if *NLOC* is not input). If *NODE* = -1 (or redefined), erase all parameters and fatigue stresses for this location.

SCFX, *SCFY*, *SCFZ*

Stress concentration factors applied to the total stresses. Factors are applied in the global X, Y, and Z directions unless the axisymmetric option of the **FSSECT** is used (i.e., *RHO* is nonzero), in which case the factors are applied in the section x, y, and z (radial, axial, and hoop) directions.

Title

User-defined title for this location (up to 20 characters).

Notes

Repeat **FL** command to define additional sets of location parameters (*MXLOC* limit), to redefine location parameters, or to delete location stress conditions.

One location must be defined for each node of interest and only one node can be associated with each location. See the **FTSIZE** command for the maximum locations (*MXLOC*) allowed. A location will be automatically defined for a node not having a location when the **FSSECT**, **FSNODE**, or **FS** command is issued. Automatically defined locations are assigned the lowest available location number, unity stress concentration factors, and no title.

Menu Paths

Main Menu>General Postproc>Fatigue>Stress Locations

FLANGE, *NLOC*, *LENG*, *MASS*, *SIF*, *FLEX*, *ARINS*, *ELEM*

Defines a flange in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

NLOC
Node where flange is to be placed (as described below). Defaults to current piping run starting point.

LENG
Length of flange (defaults to larger pipe OD).

MASS
Dry mass (weight/gravity) of flange without insulation (defaults to equivalent straight pipe mass). Note that acceleration [**ACEL**] must be nonzero for weight to be calculated.

SIF
Stress intensification factor (defaults to 1.0).

FLEX
Bending flexibility factor (defaults to 1.0).

ARINS
Insulation surface area (defaults to equivalent straight pipe insulation area). Units (length^2) must be consistent with the smallest unit of the system used (not mixed) regardless of the **PUNIT** option.

ELEM
Element number to be assigned to flange (defaults to the previous maximum element number (MAXEL) + 1).

Notes

Defines a flange (straight pipe element (PIPE16) with adjusted specifications and loadings) at a given location in a piping run. See the PREP7 **RUN** command. This command is similar to the **VALVE** command except for a different flexibility factor default. The location may be 1) between two adjacent colinear straight pipes, 2) between an adjacent straight pipe and a different piping component, or 3) at the end of a straight pipe.

For Case 1, two new nodes are generated at the ends of the flange. The two straight pipes are automatically "shortened" to meet the ends of the flange. The flange specifications and loadings are taken from the corresponding two straight pipes.

For Case 2, one new node is generated at one end of the flange. The straight pipe is automatically "shortened" to meet this end of the flange. The other end of the flange meets the other piping component. The flange specifications and loadings are taken from the straight pipe.

For Case 3, one new node is generated at the free end of the flange. The other end of the flange meets the straight pipe. The flange specifications and loadings are taken from the straight pipe.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Define Pipes>Flange

---

FLDATA, *Name*, *Label*, *Value*

Sets up a FLOTRAN analysis.

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> <> FL PP ED

Name
The name identifying the group of FLOTRAN parameters being defined or controlled on this command.

Label
The label of the specific FLOTRAN parameter being input or controlled. *Label* determines the meaning of the *Value* argument.

Value
The numeric value of an input item, the logical value of a switch (T or F, for example), or an alphanumeric label, depending on the *Label* argument.

Notes

The **FLDATA** command is used to define FLOTRAN-specific input data, solution controls, and output controls. It is valid only with the FLOTRAN CFD option.

The **FLDATA** command controls groups of FLOTRAN parameters, and the group name is input as the first argument, *Name*.

To give you the ability to link directly to the documentation on any of these groups, we have documented each group under its own name--from **FLDATA1** through **FLDATA36**. For example, documentation on **FLDATA** with the first argument set to SOLU is documented as the **FLDATA1** command. You may enter the command with either name--**FLDATA** or **FLDATA1**, and you must remember to input the appropriate first argument, as shown in the list below.

Option and Output Control Commands	
FLDATA1,SOLU	Controls which features of the solution algorithm are activated.
FLDATA2,ITER	Sets iteration and output controls for steady state analyses.
FLDATA3,TERM	Sets the convergence monitors for the pressure and temperature equations.
FLDATA4,TIME	Sets controls for transient analyses based on transient time and convergence monitors or sets time integration method.
FLDATA4A,STEP	Sets output controls for transient analyses based on number of time steps.
FLDATA5,OUTP	Sets output and storage controls.
FLDATA6,CONV	Controls the output of the convergence monitor.
Property Definition Commands	
FLDATA7,PROT	Specifies the type of fluid property.

FLDATA8,NOMI Specifies the NOMI coefficient of the fluid property equation.

FLDATA9,COF1 Specifies the COF1 coefficient of the fluid property equation.

FLDATA10,COF2 Specifies the COF2 coefficient of the fluid property equation.

FLDATA11,COF3 Specifies the COF3 coefficient of the fluid property equation.

FLDATA12,PROP Sets the property update frequency flag.

FLDATA13,VARY Sets the property variation flag.

Operating Condition Commands

FLDATA14,TEMP Specifies the reference temperature.

FLDATA15,PRES Specifies the reference pressure.

FLDATA16,BULK Specifies the bulk modulus parameter.

FLDATA17, GAMM Specifies the specific heat ratio.

Algebraic Solver Commands

FLDATA18, METH Selects the algebraic solver.

FLDATA19, TDMA Specifies the number of TDMA sweeps.

FLDATA20,SRCH Specifies the number of conjugate direction search vectors.

FLDATA20A,PGMR Specifies the amount of fill-in when preconditioning the coefficient matrix.

FLDATA20B,PBCGM Specifies the number of fill-ins for the ILU preconditioner.

FLDATA21,CONV Specifies the solver convergence criterion.

FLDATA22,MAXI Maximum number of semi-direct iterations.

FLDATA23,DELT Specifies the solver minimum normalized rate of change.

Turbulence Commands

FLDATA24,TURB Sets the turbulence model and the constants used in the Standard k-ε Model and the Zero Equation Turbulence Model.

FLDATA24A,RNGT Sets constants for the Re-Normalized Group Turbulence Model (RNG).

FLDATA24B,NKET Sets constants for the k-ε Turbulence Model due to Shih (NKE).

FLDATA24C, GIRT Sets constants for the Nonlinear Turbulence Model of Girimaji (GIR).

FLDATA24D, SZLT Sets constants for the Shih, Zhu, Lumley Turbulence Model (SZL).

FLDATA24E, SKWT Sets constants for the k-ω turbulence model.

FLDATA24F, SST1 Sets the turbulent production clip factor for the Shear Stress Transport (SST) turbulence model.

FLDATA24G, SST1 Sets constants in the k-ω regime for the Shear Stress Transport (SST) turbulence model.

FLDATA24H, SST2 Sets constants in the k-ε regime for the Shear Stress Transport (SST) turbulence model.

Stability Control Commands

FLDATA25, RELX Sets solution and property relaxation factors.

FLDATA26,STAB Sets stability controls.

Miscellaneous Commands

FLDATA27,PRIN Controls printing flags.

FLDATA28,MODR Specifies that variable results are to be replaced.

FLDATA29,MODV Re-initializes a results variable.

FLDATA30,QUAD Controls the quadrature orders.

FLDATA31,CAPP Specifies dependent variable caps.

FLDATA32,REST	Controls restart options.
FLDATA33,ADVM	Specifies the approach to discretize the advection term.
FLDATA34,MIR	Sets modified inertial relaxation factors.
FLDATA35,VFTOL	Specifies tolerances for the lower and upper bound of the volume fraction.
FLDATA36,AMBV	Specifies ambient reference values outside of the fluid for the volume of fluid (VOF) method.
FLDATA37,ALGR	Specifies segregated solution or film coefficient algorithms.
FLDATA38,MASS	Specifies the mass type for a fluid transient analysis.
FLDATA39,REMESH	Specifies remeshing parameters for transient fluid flow and fluid-solid interaction analyses.
FLDATA40,WADV	Controls activation of thermal stabilization near walls.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Advection
Main Menu>Preprocessor>FLOTRAN Set Up>Algorithm Ctrl
Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl
Main Menu>Solution>FLOTRAN Set Up>Advection
Main Menu>Solution>FLOTRAN Set Up>Algorithm Ctrl
Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl

FLDATA1, SOLU, *Label*, *Value*

Controls which features of the solution algorithm are activated.

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> <> FL PP ED

SOLU
Enter the word SOLU in this field.

FLDATA1,SOLU is the **FLDATA** command with its first argument set to SOLU. It can be entered into the program as either **FLDATA1**,SOLU,*Label*,*Value* or **FLDATA**,SOLU,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Solution algorithm choices:

FLOW
Solves the momentum and pressure equations. Defaults to TRUE (on).

TURB
Activates the turbulence model. Defaults to FALSE (off).

TEMP
Solves the temperature equation. Defaults to FALSE (off).

COMP
Uses the compressible algorithm. Defaults to FALSE (off).

SWRL
Activates the swirl option. Defaults to FALSE (off).

TRAN
Activates the transient solution algorithm. Defaults to FALSE (off).

SPEC
Activates multiple species transport. Defaults to FALSE (off).

IVSH
Activates viscous heating for incompressible flow. Defaults to FALSE (off).

VOF
Activates volume of fluid method. Defaults to FALSE (off).

SFTS
Activates surface tension. Defaults to FALSE (off).

ALE
Activates Arbitrary Lagrangian-Eulerian formulation. Defaults to FALSE (off).

RDSF
Activates the surface-to-surface radiation solution. Defaults to TRUE (on).

Value
Value controlling *Label*:

TRUE or T
Turn this feature on.

FALSE or F
Turn this feature off.

Notes

Repeat command to set each *Label* as required.

The analyst must choose the appropriate features, as FLOTRAN will not determine, for example, whether or not the case is turbulent or whether or not the compressible algorithm is appropriate.

SWRL should only be activated for axisymmetric cases when there is a velocity component normal to the axisymmetric plane.

Generally, the use of the compressible algorithm (COMP) is not warranted for Mach numbers less than 0.3. Density may be assumed to vary via the ideal gas law without activating the compressible option.

If fluid properties are not a function of temperature in non-adiabatic flow problems, it is not necessary to activate the flow (FLOW) and temperature (TEMP) solutions together. First solve the flow problem and then restart to solve the temperature equation.

For compressible flow, FLOTRAN automatically includes viscous heating.

When the Arbitrary Lagrangian-Eulerian (ALE) formulation is on , FLOTRAN allows the fluid nodes to move in a manner that satisfies the displacement boundary conditions.

See also the **FLDATA2**,ITER, **FLDATA3**,TERM, **FLDATA4**,TIME, **FLDATA4A**,STEP, **FLDATA5**,OUTP, and **FLDATA6**,CONV commands for other Solution and Output Controls.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Solution Options
Main Menu>Solution>FLOTRAN Set Up>Solution Options

FLDATA2, ITER, *Label*, *Value*

Sets iteration and output controls for steady state analyses.

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> <> FL PP ED

ITER
Enter the word ITER in this field.

FLDATA2,ITER is the **FLDATA** command with its first argument set to ITER. It can be entered into the program as either **FLDATA2**,ITER,*Label*,*Value* or **FLDATA**,ITER,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Type of iteration control (dictates the meaning of *Value*):

EXEC
Value is the number of global iterations, defined as the sequential solution of the governing equations of all the features activated, to be performed during this execution of FLOTRAN if the case is steady state. Defaults to *Value* = 10.

APPE
Value is the number of steady state global iterations between appends to the results file (**Jobname.RFL**). This feature is used to ensure that restarts can be made from earlier points if unforeseen difficulties occur before the end of the analysis. The default value implies that results will be saved only for the initial (0^{th}) iteration and the final global iteration. Saving many intermediate results can produce a large results file. Defaults to *Value* = 0 (implies **FLDATA2**,ITER,EXEC).

If you are creating domain files (**jobname.pv_0000*n***) for use by ICEM CFD's PV3 postprocessing visualization tool, this sets the frequency with which domain files are written. See **FLDATA5** for more details.

OVER
Value is the number of steady state global iterations between overwrites of a temporary set of results in the results file (**Jobname.RFL**). (This temporary set of results will itself be overwritten by the set stored for Label = APPE or at the end of the run.) OVER enables the user to retain the most up-to-date set of results without having the results file continue to grow in size. Defaults to *Value* = 0 (no overwrites will be made).

Value
Number of iterations for *Label* above.

Notes

Repeat command to set each *Label* as required.

Sets the number of global iterations to control the length of execution for steady state analyses, the frequency with which results are added to the results file (**Jobname.RFL**), and the frequency with which results are over-

written in the results file. Also sets the frequency with which the domain files, used by the ICEM PV3 postprocessing visualization tool, are written. For more information about domain files, see the **FLDATA5**,OUTP,DOMA command.

See also the **FLDATA1**,SOLU, **FLDATA3**,TERM, **FLDATA4**,TIME, **FLDATA4A**,STEP, **FLDATA5**,OUTP, and **FLDATA6**,CONV commands for other Solution and Output Controls.

This command is accessible in the menu if **FLDATA1**,SOLU,TRAN,FALSE has been issued (default).

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl
Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl

FLDATA3, TERM, *Label*, *Value*

Sets the convergence monitors for the degree of freedom set.

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> <> FL PP ED

TERM
Enter the word TERM in this field.

FLDATA3,TERM is the **FLDATA** command with its first argument set to TERM. It can be entered into the program as either **FLDATA3**,TERM,*Label*,*Value* or **FLDATA**,TERM,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Type of convergence monitor:

PRES
Steady-state run will not terminate until the convergence monitor for pressure falls below *Value*, unless the specified number of global iterations has been executed. Defaults to *Value* = 1.0 x 10^{-8}.

TEMP
Steady-state run (with the temperature equation solution activated) will not terminate until the convergence monitor for temperature falls below *Value*, unless the specified number of global iterations has been executed. Defaults to *Value* = 1.0 x 10^{-8}.

VX
Steady-state run will not terminate until the convergence monitor for X velocity component falls below *Value*, unless the specified number of global iterations has been executed. Defaults to *Value* = 1.0 x 10^{-2}.

VY
Steady-state run will not terminate until the convergence monitor for Y velocity component falls below *Value*, unless the specified number of global iterations has been executed. Defaults to *Value* = 1.0 x 10^{-2}.

VZ
Steady-state run will not terminate until the convergence monitor for Z velocity component falls below *Value*, unless the specified number of global iterations has been executed. Defaults to *Value* = 1.0 x 10^{-2}.

ENKE
Steady-state run will not terminate until the convergence monitor for turbulent kinetic energy falls below *Value*, unless the specified number of global iterations has been executed. Defaults to *Value* = 1.0 x 10^{-2}.

ENDS
Steady-state run will not terminate until the convergence monitor for turbulence dissipation falls below *Value*, unless the specified number of global iterations has been executed. Defaults to *Value* = 1.0 x 10^{-2}.

Value
Value of convergence monitor criterion, above.

Notes

Repeat command to set each *Label* as required.

The convergence monitors appear as output for each degree of freedom for each global iteration. The value is calculated for each degree of freedom individually by summing the absolute value of the change in solution between global iterations for all the nodes and dividing it by the sum of the absolute values of the solution for all the nodes. It is an approximation of the normalized rate of change of the solution between global iterations.

All specified criteria must be met before the case is terminated.

If a termination criterion for a specific label is set negative, the termination check is ignored for that particular DOF.

See also the **FLDATA1**,SOLU, **FLDATA2**,ITER,**FLDATA4**,TIME, **FLDATA4A**,STEP, **FLDATA5**,OUTP, and **FLDATA6**,CONV commands for other Solution and Output Controls.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl
Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl

FLDATA4, TIME, *Label*, *Value*

Sets controls for transient analyses based on transient time and convergence monitors or sets time integration method.

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> <> FL PP ED

TIME
Enter the word TIME in this field.

FLDATA4,TIME is the **FLDATA** command with its first argument set to TIME. It can be entered into the program as either **FLDATA4**,TIME,*Label*,*Value* or **FLDATA**,TIME,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Type of transient analysis control or time integration method (dictates the meaning of *Value*):

STEP

Value controls the time step size. If *Value* is greater than zero, *Value* is the time step size. If *Value* is less than zero, the program chooses the time step, according to the following (*Value* defaults to -1):

1

The resulting time step will be small enough to prevent the passage of an arbitrarily small fluid parcel through more than one element length in a single time step.

2

This choice is only applicable for compressible analyses, and is equivalent to the Courant limit. The time step is small enough to prevent a pressure signal from propagating through more than one element during a single time step.

3

This choice is only applicable for compressible analyses. The smaller of the two time steps computed for options -1 and -2 is used.

4

This choice is applicable to "conduction only" cases (**FLDATA1**,SOLU,FLOW,F). The resulting time step prevents an arbitrarily small "parcel of heat" from diffusing or conducting through more than one element within a time step.

ISTEP

Value is the time step size for the first time step in an analysis using a FLOTRAN-calculated time step size (STEP = -1 through -4).

BC

Value is a flag indicating whether a transient boundary condition should be applied as a step change (when *Value* = 0) or as a linear ramp (*Value* = 1). This label is analogous to the ANSYS command **KBC**, except that the default for FLOTRAN is a step change. Tabular boundary conditions do not support ramping and instead apply their full value regardless of the **KBC** setting.

NUMB

Value is the number of time steps which will be executed unless the transient end time has been encountered first.

GLOB

Value is the number of global iterations per time step. The user may elect to set this to a high value (typically between 30 and 50), and expect the time step to converge to the convergence criterion set by the **FLDATA3**,TERM command before completing *Value* iterations. Default to *Value* = 10.

TEND

Value is the transient end time. This time is used for the calculation of ramped transient boundary conditions. The case will execute until this end time if the number of time steps specified is large enough. Defaults to *Value* = $1.0 \times 10^{+6}$.

APPE

Value is the results output frequency based on transient time. Every *Value* seconds in the transient, results are written to the **Jobname.RFL** file. If you are creating domain files (**jobname.pv_0000*n***) for use by ICEM CFD's PV3 postprocessing visualization tool, this sets the frequency with which domain files are written. See **FLDATA5** for more details. Defaults to *Value* = $1.0 \times 10^{+6}$.

SUMF

Value is the output summary frequency based on transient time. Every *Value* seconds in the transient, a results summary is written to the **Jobname.PFL** file.

Note — A summary will always be written when a set of results is written to the **Jobname.RFL** file. Defaults to Value = 1.0 x 10^{+6}.

OVER
Value is the time interval between overwrites of the temporary set of results in the results file (**Jobname.RFL**). This feature enables the user to retain the most up-to-date set of results without having the results file continue to grow in size. Defaults to *Value* = 0 (no overwrites will be made).

PRES
If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for pressure falls below *Value*. Defaults to *Value* = 1.0 x 10^{-6}.

TEMP
If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for temperature falls below *Value*. Defaults to *Value* = 1.0 x 10^{-6}.

VX
If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for X velocity component falls below *Value*. Defaults to *Value* = 1.0 x 10^{-2}.

VY
If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for Y velocity component falls below *Value*. Defaults to *Value* = 1.0 x 10^{-2}.

VZ
If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for Z velocity component falls below *Value*. Defaults to *Value* = 1.0 x 10^{-2}.

ENKE
If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for turbulent kinetic energy falls below *Value*. Defaults to *Value* = 1.0 x 10^{-2}.

ENDS
If the maximum number of global iterations per time step has not been reached, the time step will terminate if the convergence monitor for turbulence dissipation falls below *Value*. Defaults to *Value* = 1.0 x 10^{-2}.

NTVF
Value controls the time stepping strategy for Volume of Fluid (VOF) Advection. The value gives the number of time steps used in the VOF Advection for each time step in the FLOTRAN solution. Defaults to 1.

METH
Value specifies the time integration method. Allowable choices for *Value* are:

BACK
Backward method (default).

NEWM
Newmark time integration method.

DELT

Newmark parameter to control integration accuracy and stability. It must be $\geq$ 0.5. Defaults to 0.5 (2nd order accuracy). If it is larger than 0.5, numerical damping is introduced and the calculation is more stable. The recommended maximum value is 0.6.

Value

Value as described for *Label* above.

Notes

Repeat command to set each *Label* as required.

All specified criteria must be met before the time step is terminated.

If a termination criterion for a specific label is set negative, the termination check is ignored for that particular DOF.

See the **FLDATA4A**,STEP command to specify output controls based on the number of time steps.

See also the **FLDATA1**,SOLU, **FLDATA2**,ITER, **FLDATA3**,TERM, **FLDATA5**,OUTP, and **FLDATA6**,CONV commands for other Solution and Output Controls.

This command is accessible in the menu if **FLDATA1**,SOLU,TRAN,TRUE has been issued.

For ramped loading (**FLDATA4**,TIME,BC,1), when a load is applied for the first time, it is interpolated from zero to the value of the current load step, and not from the initial condition or value of the DOF from the previous load step.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl
Main Menu>Preprocessor>FLOTRAN Set Up>Transient Ctrl>Time Integration Meth
Main Menu>Preprocessor>FLOTRAN Set Up>VOF Environment>Time Stepping
Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl
Main Menu>Solution>FLOTRAN Set Up>Transient Ctrl>Time Integration Meth
Main Menu>Solution>FLOTRAN Set Up>VOF Environment>Time Stepping

---

FLDATA4A, STEP, *Label*, *Value*

Sets controls for transient analyses based on the number of time steps.

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> <> FL PP ED

STEP

Enter the word STEP in this field.

FLDATA4A,STEP is the **FLDATA** command with its first argument set to STEP. It can be entered into the program as either **FLDATA4A**,STEP,*Label*,*Value* or **FLDATA**,STEP,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label

Type of transient analysis control (dictates the meaning of *Value*):

APPE
Value is the results output frequency based on the number of time steps. Every *Value* time steps (substeps), results are written to the **Jobname.RFL** file. If you are creating domain files (**jobname.pv_0000*n***) for use by ICEM CFD's PV3 postprocessing visualization tool, this sets the frequency with which domain files are written. See **FLDATA5** for more details. Defaults to *Value* = 10.

SUMF
Value is the output summary frequency based on the number of time steps. Every *Value* time steps (substeps), a results summary is written to the **Jobname.PFL** file.

Note — A results summary will be produced automatically when the results of a time step are written to the **Jobname.RFL** file. Defaults to *Value* = 10.

OVER
Value is the number of time steps between overwrites of the temporary set of results in the results file (**Jobname.RFL**). This feature enables the user to retain the most up-to-date set of results without having the results file continue to grow in size. Defaults to *Value* = 0 (no overwrites will be made).

Notes

Repeat command to set each *Label* as required.

See the **FLDATA4**,TIME command to specify output controls based on transient time.

See also the **FLDATA1**,SOLU, **FLDATA2**,ITER, **FLDATA3**,TERM, **FLDATA5**,OUTP, and **FLDATA6**,CONV commands for other solution and output controls.

This command is accessible in the menu if **FLDATA1**,SOLU,TRAN,TRUE has been issued.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl
Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl

FLDATA5, OUTP, *Label*, *Value*

Sets output and storage controls.

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> <> FL PP ED

OUTP
Enter the word OUTP in this field.

FLDATA5,OUTP is the **FLDATA** command with its first argument set to OUTP. It can be entered into the program as either **FLDATA5**,OUTP,*Label*,*Value* or **FLDATA**,OUTP,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Output and storage controls (dictates the meaning of *Value*):

SUMF

Value is the number of global iterations between output of results summaries. The results summary, output to the file **Jobname.PFL**, contains a tabulation of the maximum, minimum and average values of each degree of freedom. It also contains the flow rates, pressures, and temperatures at all the inlets and outlets. Defaults to *Value* = 10.

DEBG

Value sets the debug file printout level. The debug file **Jobname.DBG** contains information on the behavior of the semi-direct solvers used in the solution of the equations. Allowable choices for *Value* are (defaults to 1):

0

No information.

1

Initial and final information for each global iteration.

2

Complete information for each global iteration.

3

Produces printout of global coefficient matrices (not recommended, since the files will become very large).

4

Produces printout of matrices and additional values at every global iteration (not recommended, because files are usually very large).

RESI

Value is a flag controlling whether or not the nodal residual file **Jobname.RDF** is written. The nodal residual file contains the nodal residuals corresponding to the current solution (n-1st global iteration), and the new coefficient matrices and forcing function (nth global iteration). It indicates whether or not the solution is oscillating between global iterations on a nodal basis. The file produced is a text file (**Jobname.RDF**) that is read with the **FLREAD** command and that contains the residuals for the degrees of freedom (VX, VY, VZ, PRES, ENKE, ENDS and TEMP) for each node. Allowable values for *Value* are T (writes the file) or F (does not write the file). Defaults to *Value* = F.

DENS

Value controls the storage of the laminar density. This and the following labels are provided to control the size of the **Jobname.RFL** file (see notes below). The choices are *Value* = T (stores this component) or *Value* = F (does not store this component). Defaults to *Value* = T.

VISC

Value controls the storage of the laminar viscosity. Defaults to *Value* = T.

COND

Value controls the storage of the laminar conductivity. Defaults to *Value* = T.

EVIS

Value controls the storage of the effective viscosity. Defaults to *Value* = T.

ECON

Value controls the storage of the effective conductivity. Defaults to *Value* = T.

TTOT

Value controls the storage of the total temperature. Defaults to *Value* = T.

HFLU

Value controls the storage of the heat flux. Defaults to *Value* = T.

HFLM
Value controls the storage of the heat transfer (film) coefficient. Defaults to *Value* = T.

SPHT
Value controls the storage of the specific heat. Defaults to *Value* = F.

STRM
Value controls the storage of the stream function (2-D). Defaults to *Value* = T.

MACH
Value controls the storage of the Mach number. Defaults to *Value* = T.

PTOT
Value controls the storage of the total (stagnation) pressure. Defaults to *Value* = T.

PCOE
Value controls the storage of the pressure coefficient. Defaults to *Value* = T.

YPLU
Value controls the storage of the Y+ turbulence quantity. Defaults to *Value* = F.

TAUW
Value controls the storage of the shear stress at the wall. Defaults to *Value* = F.

SFTS
Value controls the storage of the surface tension coefficient. Defaults to *Value* = F.

RDFL
Value controls the storage of the radiation heat flux. Defaults to *Value* = F.

LMD*n*
Laminar mass diffusion coefficient for species *n*, where *n* = 1 to 6.

EMD*n*
Effective mass diffusion coefficient for species *n*, where *n* = 1 to 6.

DOMA
Value controls whether domain files are written. For transient analysis, if *Value*= T, domain files are written at the time frequency specified by the **FLDATA4** command or the step frequency set by the **FLDATA4A** command. For steady state analysis, the files are written as specified by the **FLDATA2** command. Files are named **jobname.pv_00001**, **jobname.pv_00002**, etc., corresponding to the first time increment, second time increment, and so forth. Result set numbers (First, Last, Next, Max, etc.) are preserved to ensure that any domain files created during a previous analysis are either deleted or preserved appropriately.

Setting *Value*= F turns domain file writing off (the default state).

DRAD
VALUE controls debug level of convergence monitor for surface-to-surface radiation method. Allowable choices for value are (defaults to 0):

0
No information (default).

1
Final convergence information.

2
Complete information for each global iteration.

Value
Value as described above.

Notes

Repeat command to set each *Label* as required.

SUMF, DEBG, and RESI control output to files other than the nodal results file (**Jobname.RFL**). DOMA controls output to domain files (**Jobname.pv0000*n***), which are for use with the ICEM CFD PV3 postprocessing visualization tool.

The remaining *Label* labels are provided in the event the user needs to control the size of the **Jobname.RFL** file. This may be the case when the results of many load steps or time steps are being stored for large models. The choices are T (stores this component) or F (does not store this component). Since the controls should only be set to F if the file size is a problem, most default values are T, and values of listed parameters are stored for every node.

FLOTRAN only allows non-storage of variables which it can calculate from some other means. If the value is not stored, it is replaced by calculations identical to those of the normal solution algorithm. The difference is that the new properties calculated in the absence of storage have not been relaxed as those in storage would have been. See **FLDATA25**,RELX for a description of relaxation.

Laminar properties such as density, viscosity, and thermal conductivity are either constant or a function of temperature. In the former case, the constant value can be obtained from the input. In the case of temperature dependent properties, the correct values are obtained from the temperature field (always stored if the temperature equation solution was activated [**FLDATA1**,SOLU,TEMP,TRUE]). The newly calculated value is stored as calculated at the initialization of the load step, whereas the value obtained from storage would have undergone relaxation.

If the effective viscosity and effective thermal conductivity are not stored, upon restart they will be calculated from the existing field of turbulent kinetic energy, turbulent kinetic energy dissipation rate, and the density. Again, this is similar to the usual calculation.

See also the **FLDATA1**,SOLU, **FLDATA2**,ITER, **FLDATA3**,TERM, **FLDATA4**,TIME, **FLDATA4A**,STEP, and **FLDATA6**,CONV commands for other Solution and Output Controls.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Additional Out>Print Controls
Main Menu>Preprocessor>FLOTRAN Set Up>Additional Out>Residual File
Main Menu>Preprocessor>FLOTRAN Set Up>Additional Out>RFL Out Derived
Main Menu>Preprocessor>FLOTRAN Set Up>Additional Out>RFL Prop Based
Main Menu>Preprocessor>FLOTRAN Set Up>Execution Ctrl
Main Menu>Solution>FLOTRAN Set Up>Additional Out>Print Controls
Main Menu>Solution>FLOTRAN Set Up>Additional Out>Residual File
Main Menu>Solution>FLOTRAN Set Up>Additional Out>RFL Out Derived
Main Menu>Solution>FLOTRAN Set Up>Additional Out>RFL Prop Based
Main Menu>Solution>FLOTRAN Set Up>Execution Ctrl

FLDATA6, CONV, *Label*, *Value*

Controls the output of the convergence monitor.

PREP7: FLOTRAN Options

MP <> <> <> <> <> <> <> FL PP ED

CONV
Enter the word CONV in this field.

FLDATA6,CONV is the **FLDATA** command with its first argument set to CONV. It can be entered into the program as either **FLDATA6**,CONV,*Label*,*Value* or **FLDATA**,CONV,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Format and frequency controls (dictates the meaning of *Value*):

OUTP
Value controls which display format is used. *Value* can be LAND, BLOC, or BNOW, as described below (defaults to BNOW).

ITER
If OUTP *Value* is LAND or BLOC, *Value* is the number of iterations for which the convergence monitor information is stored in memory before being printed. Defaults to *Value* = 1.

Value
Value as described for *Label* above:

LAND
All DOF are output in landscape mode (valid only if *Label* = OUTP).

BLOC
All DOF are output in block mode (valid only if *Label* = OUTP).

BNOW
All DOF are presented in block format immediately as iterations are completed (valid only if *Label* = OUTP).

n
Number of iterations (valid only if *Label* = ITER).

Notes

The convergence monitor output will appear on the screen during an interactive FLOTRAN analysis as well as in the printed output. OUTP controls what display format is used, and ITER controls how many iterations are stored and displayed.

See also the **FLDATA1**,SOLU, **FLDATA2**,ITER, **FLDATA3**,TERM, **FLDATA4**,TIME, **FLDATA4A**,STEP, and **FLDATA5**,OUTP commands for other Solution and Output Controls.

The choices for OUTP are LANDscape mode, BLOCk mode, or the "block now" mode, BNOW. If OUTP is set to BNOW, convergence monitor information is not stored in the memory, but is presented immediately as it is calculated. If OUTP is set to either LAND or BLOC, information is first stored for *n* (ITER) iterations before being printed.

In the following illustrations, "x" denotes convergence monitor numbers greater than zero:

Landscape Mode

Iter<	1	2	3	4	5	6
VX	x	x	x	x	x	x
VY	x	x	x	x	x	x
VZ	x	x	x	x	x	x
PRES	x	x	x	x	x	x
ENKE	x	x	x	x	x	x
ENDS	x	x	x	x	x	x
TEMP	x	x	x	x	x	x

Block Mode

Iter	VX	VY	VZ	PRES	ENKE	ENDS	TEMP
^							
1	x	x	x	x	x	x	X
2	x	x	x	x	x	x	X
3	x	x	x	x	x	x	X
4	x	x	x	x	x	x	X
5	x	x	x	x	x	x	X
6	x	x	x	x	x	x	X

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Additional Out>Print Controls
Main Menu>Solution>FLOTRAN Set Up>Additional Out>Print Controls

---

FLDATA7, PROT, *Label*, *Value*

Specifies the type of fluid property.

PREP7: FLOTRAN Property

MP <> <> <> <> <> <> <> FL PP ED

PROT
Enter the word PROT in this field.

FLDATA7,PROT is the **FLDATA** command with its first argument set to PROT. It can be entered into the program as either **FLDATA7**,PROT,*Label*,*Value* or **FLDATA**,PROT,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Fluid property being typed:

DENS
Density (*Value* defaults to CONSTANT).

VISC
Viscosity (*Value* defaults to CONSTANT).

COND
Thermal conductivity (*Value* defaults to CONSTANT).

SPHT
Specific heat (*Value* defaults to CONSTANT).

SFTS
Surface tension coefficient (*Value* defaults to CONSTANT).

WSCA
Wall static contact angle. (*Value* defaults to CONSTANT).

Value
Fluid property type.

You can enter one of the values shown below, or if your fluid properties can be expressed as a table, you can enter a table name. To enter a table name, you must first define a TABLE type array parameter using the ***DIM** command. Note that you must enclose the table name in % symbols in the **FLDATA7** command line (e.g., **FLDATA7**,PROT,DENS,%tabname%). For more information on defining tables, see TABLE Type Array Parameters in the *ANSYS APDL Programmer's Guide*.

CONSTANT
Constant properties. The **FLDATA8**,NOMI command must be used to specify nominal properties.

GAS
Gas properties. The **FLDATA8**,NOMI command must be used to specify nominal properties. **FLDATA9**,COF1, **FLDATA10**,COF2, and possibly **FLDATA11**,COF3, must also be used.

LIQUID
Liquid properties. The **FLDATA8**,NOMI command must be used to specify nominal properties. **FLDATA9**,COF1, **FLDATA10**,COF2, and possibly **FLDATA11**,COF3 must be used.

TABLE
Indicates that a table of property values and corresponding temperature values are input using the **MPDATA** and **MPTEMP** commands. You cannot use this option with the %table% method described above.

POWL
This choice for viscosity type activates the Power Law viscosity model, which is non-Newtonian. (For a description of the model see the *ANSYS, Inc. Theory Reference*. The *ANSYS Fluids Analysis Guide* explains how to use the model.) The Power Law model requires you to specify four coefficients via the **FLDATA8**,NOMI,VISC command, the **FLDATA9**,COF1,VISC command, the **FLDATA10**,COF2,VISC command, and the **FLDATA11**,COF3,VISC command.

CARR
This choice for viscosity type activates the Carreau viscosity model, which in non-Newtonian. (For a description of this model, see the *ANSYS, Inc. Theory Reference*. The *ANSYS Fluids Analysis Guide* explains how to use the non-Newtonian viscosity models.) The Carreau model requires you to specify four coefficients via the **FLDATA8**, NOMI,VISC command, the **FLDATA9**,COF1,VISC command, the **FLDATA10**,COF2,VISC command and the **FLDATA11**,COF3,VISC command.

BING
This choice for viscosity type activates the Bingham viscosity model, which is non-Newtonian. (For a description of this model, see the *ANSYS, Inc. Theory Reference*. The *ANSYS Fluids Analysis Guide* explains how to use the non-Newtonian viscosity models.) The Bingham model requires you to specify three coefficients, using the **FLDATA8**,NOMI,VISC command, the **FLDATA9**,COF1,VISC command, and the **FLDATA10**,COF2,VISC command.

USRV
This choice for viscosity activates the user-programmable subroutine, USERVISLAW. In this routine, you can define your own constitutive relationship between viscosity and other variables such as position,

time, temperature, pressure, velocity, velocity gradients, etc. For details, see the *ANSYS, Inc. Theory Reference*, the *ANSYS Fluids Analysis Guide*, and the *Guide to ANSYS User Programmable Features*

The USERVISLAW subroutine uses the four coefficients you specify via the **FLDATA8**,NOMI,VISC command, the **FLDATA9**,COF1,VISC command, the **FLDATA10**,COF2,VISC command, and the **FLDATA11**,COF3,VISC command.

AIR
Air properties in units of meter-kg-sec.

AIR_B
Air properties in units of meter-kg-sec, and the pressure is set to the reference pressure for the evaluation of density.

AIRSI
Air properties in units of meter-kg-sec.

AIRSI_B
Air properties in units of meter-kg-sec, and the pressure is set to the reference pressure for the evaluation of density.

AIRCM
Air properties in units of cm-g-sec.

AIRCM_B
Air properties in units of cm-g-sec, and the pressure is set to the reference pressure for the evaluation of density.

AIRMM
Air properties in units of mm-g-sec.

AIRMM_B
Air properties in units of mm-g-sec, and the pressure is set to the reference pressure for the evaluation of density.

AIRFT
Air properties in units of fl-slug-sec.

AIRFT_B
Air properties in units of fl-slug-sec, and the pressure is set to the reference pressure for the evaluation of density.

AIRIN
Air properties in units of in-(lbf-s**2/in)-sec (results in units of psi for pressure).

AIRIN_B
Air properties in units of in-(lbf-s**2/in)-sec (results in units of psi for pressure), and the pressure is set to the reference pressure for the evaluation of density.

CMIX
The property is the mass fraction average of the individual species property. You can use this option only if the species are defined.

USER
Use one of the following user-programmable subroutines to define the property: UserDens to define density, UserVisLaw to define viscosity, UserSpht to define specific heat, UserCond to define conductivity, or UserSfts to define surface tension.

CGAS
Available only for density. The density is calculated using the ideal gas law, with the molecular weight calculated from the mass fraction average of the individual species molecular weights. You can only use this option if species are defined.

Notes

This command is used to specify the fluid property type (constant, gas, liquid, air) for the density, viscosity, and (if required) thermal conductivity and specific heat properties. Repeat the **FLDATA7** command for each property as required.

The choice of fluid property type implies the use of one of the following equations. In all cases the program assumes that consistent units are being used. The value of unity for the gravitational constant g_c implies a consistent set of units.

If the property type is CONSTANT, then the equations used are as follows:

DENS = NOMI
VISC = NOMI
COND = NOMI
SPHT = NOMI

If the property type is LIQUID, then Sutherland's liquid law is used for the viscosity and conductivity and a second order polynomial as a function of temperature is used for density:

$$DENS = NOMI + COF2*(T-COF1) + COF3*(T-COF1)^2$$

$$VISC = NOMI*EXP[(COF2*(1/T-1/COF1) + COF3*(1/T-1/COF1)^2]$$

$$COND = NOMI*EXP[(COF2*(1/T-1/COF1) + COF3*(1/T-1/COF1)^2]$$

If the type is GAS, Sutherland's law for gases is used for conductivity and viscosity and the ideal gas law is used for the density:

$$DENS = NOMI * (P/COF2) / (T/COF1)$$

$$VISC = NOMI * (T/COF1)^{1.5} * (COF1 + COF2)/(T + COF2)$$

$$COND = NOMI * (T/COF1)^{1.5} * (COF1 + COF2)/(T + COF2)$$

For *Label* = SFTS, CONSTANT, LIQUID, and USER are the only valid property types. If the property type is CONSTANT, the equation is:

$$SFTS = NOMI$$

If the property type is LIQUID, the equation is:

$$SFTS = NOMI + COF2*(T-COF1) + COF3*(T-COF1)^2$$

For *Label* = WSCA, CONSTANT is the only valid property type and the equation is:

$$WSCA = NOMI$$

In each case, the value of NOMI is input with the **FLDATA8**,NOMI command, COF1 with **FLDATA9**,COF1, COF2 with **FLDATA10**,COF2, and COF3 with **FLDATA11**,COF3. No defaults are assumed.

For types other than GAS, LIQUID, CONSTANT, AIR or AIR_B, the values of constants in the evaluation of properties will be obtained from the file FLOPRP.ANS, where the data for AIR resides. See the *ANSYS Fluids Analysis Guide* for information on how to put data into the FLOPRP.ANS file.

See also the **FLDATA8**,NOMI, **FLDATA9**,COF1, **FLDATA10**,COF2, **FLDATA11**,COF3, **FLDATA12**,PROP, and **FLDATA13**,VARY commands for other Fluid Property Definition commands.

This command is accessed in the menu as **FLDATA12**,PROP.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties
Main Menu>Preprocessor>FLOTRAN Set Up>Table Props
Main Menu>Solution>FLOTRAN Set Up>Fluid Properties
Main Menu>Solution>FLOTRAN Set Up>Table Props

FLDATA8, NOMI, *Label*, *Value*

Specifies the NOMI coefficient of the fluid property equation.

PREP7: FLOTRAN Property

MP <> <> <> <> <> <> <> FL PP ED

NOMI
Enter the word NOMI in this field.

FLDATA8,NOMI is the **FLDATA** command with its first argument set to NOMI. It can be entered into the program as either **FLDATA8**,NOMI,*Label*,*Value* or **FLDATA**,NOMI,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Fluid property NOMI is being defined for:

DENS
Density.

VISC
Viscosity.

COND
Thermal conductivity.

SPHT
Specific heat.

SFTS
Surface tension coefficient.

WSCA
Wall static contact angle (*Value* defaults to 90).

Value
Enter the value for NOMI.

Notes

Specifies the value NOMI as described on the **FLDATA7**,PROT command. NOMI is the constant fluid property value, or is the value of the property at the specified temperature COF1 (or simply the value of a coefficient). NOMI is only valid for property types GAS, LIQUID, and CONSTANT [**FLDATA7**,PROT]. If the property is a gas or liquid, **FLDATA9**,COF1, **FLDATA10**,COF2, and possibly **FLDATA11**,COF3, must also be used.

If the label is VISC, the **FLDATA7**,PROT command you enter determines what NOMI is. For example, if you issue the **FLDATA7**,PROT,VISC,POWL command, NOMI is the nominal viscosity for the Power Law model. If you issue the **FLDATA7**,PROT,VISC,BING command, the NOMI is the plastic viscosity for the Bingham model. If you issue the **FLDATA7**,PROT,VISC,CARR command, the NOMI is the zero shear rate viscosity for the Carreau model. If you issue the **FLDATA7**,PROT,USRV command, the NOMI is a coefficient available in USERVISLAW, the user-programmable subroutine for viscosity models.

See also the **FLDATA12**,PROP and **FLDATA13**,VARY commands for other Fluid Property Definition commands.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties
Main Menu>Solution>FLOTRAN Set Up>Fluid Properties

FLDATA9, COF1, *Label*, *Value*

Specifies the COF1 coefficient of the fluid property equation.

PREP7: FLOTRAN Property

MP <> <> <> <> <> <> <> FL PP ED

COF1
Enter the word COF1 in this field.

FLDATA9,COF1 is the **FLDATA** command with its first argument set to COF1. It can be entered into the program as either **FLDATA9**,COF1,*Label*,*Value* or **FLDATA**,COF1,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Fluid property being described:

DENS
Density.

VISC
Viscosity.

COND
Thermal conductivity.

SPHT
Specific heat.

SFTS
Surface tension coefficient.

Value
Absolute temperature at which the *Label* property has the value NOMI (defaults to 0.0) or simply a coefficient.

Notes

Specifies the value of absolute temperature associated with the property value set be **FLDATA8**,NOMI, as described on the **FLDATA7**,PROT command, or simply a coefficient. Setting the value of COF1 to zero results in a constant property for liquids or gas. If the property is a gas or liquid, **FLDATA8**,NOMI, **FLDATA10**,COF2, and possibly **FLDATA11**,COF3, must also be used.

If the label is VISC and you issue the **FLDATA7**,PROT,VISC,POWL command, the COF1 value is the cutoff shear rate for the Power Law Model. If you issue the **FLDATA7**,PROT,VISC,BING command, COF1 is the plastic stress value for the Bingham model. If you issue the **FLDATA7**,PROT,VISC,CARR command, the COF1 value is the infinite shear rate viscosity for the Carreau model. If you issue the **FLDATA7**,PROT,VISC,USRV command, COF1 is a coefficient available in the USERVISLAW subroutine.

If the label is SPHT, the COF1 value is available in the user-programmable subroutine UserSpht.

See also the **FLDATA12**,PROP, and **FLDATA13**,VARY commands for other Fluid Property Definition commands.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties
Main Menu>Solution>FLOTRAN Set Up>Fluid Properties

FLDATA10, COF2, *Label*, *Value*

Specifies the COF2 coefficient of the fluid property equation.

PREP7: FLOTRAN Property

MP <> <> <> <> <> <> <> FL PP ED

COF2
Enter the word COF2 in this field.

FLDATA10,COF2 is the **FLDATA** command with its first argument set to COF2. It can be entered into the program as either **FLDATA10**,COF2,*Label*,*Value* or **FLDATA**,COF2,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Fluid property being described:

DENS
Density.

VISC
Viscosity.

COND
Thermal conductivity.

SPHT
Specific heat.

SFTS
Surface tension coefficient.

Value
Coefficient COF2 (defaults to 0.0).

Notes

Specifies the value COF2 as described on the **FLDATA7**,PROT command.

If the label is VISC and you issue the **FLDATA7**,PROT,VISC,POWL command, the COF2 value is the consistency coefficient for the Power Law Model. If you issue the **FLDATA7**,PROT,VISC,BING command, COF2 is the Newtonian viscosity for the Bingham model. If you issue the **FLDATA7**,PROT,VISC,CARR command, the COF2 value is the time constant for the Carreau model. If you issue the **FLDATA7**,PROT,VISC,USRV command, COF2 is a coefficient available in the USERVISLAW subroutine.

For viscosity, liquid, gas, or thermal conductivity, COF2 is a Sutherland's law constant. For liquid density, it is a polynomial coefficient. For gas density, it is the pressure in the nominal ideal gas law constant evaluation. If the property is a gas or liquid, **FLDATA8**,NOMI, **FLDATA9**,COF1, and possibly **FLDATA11**,COF3, must also be used.

See also the **FLDATA12**,PROP, and **FLDATA13**,VARY commands for other Fluid Property Definition commands.

If the label is SPHT, the COF2 value is available in the user-programmable subroutine UserSpht.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties
Main Menu>Solution>FLOTRAN Set Up>Fluid Properties

FLDATA11, COF3, *Label*, *Value*

Specifies the COF3 coefficient of the fluid property equation.

PREP7: FLOTRAN Property

MP <> <> <> <> <> <> <> FL PP ED

COF3
Enter the word COF3 in this field.

FLDATA11,COF3 is the **FLDATA** command with its first argument set to COF3. It can be entered into the program as either **FLDATA11**,COF3,*Label*,*Value* or **FLDATA**,COF3,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Fluid property being described:

DENS
Density.

VISC
Viscosity.

COND
Thermal conductivity.

SPHT
Specific heat.

SFTS
Surface tension coefficient.

Value
Coefficient COF3 (defaults to 0.0).

Notes

Specifies the value COF3 as described on the **FLDATA7**,PROT command.

If the label is VISC and you issue the **FLDATA7**,PROT,VISC,POWL command, the COF3 value is the power for the Power Law Model. COF3 is not used in the Bingham model. If you issue the **FLDATA7**,PROT,VISC,CARR command, the COF3 value is the power for the Carreau model. If you issue the **FLDATA7**,PROT,VISC,USRV command, COF3 is a coefficient available in the USERVISLAW subroutine.

For the viscosity or thermal conductivity of a liquid, COF3 is a Sutherland's law constant. For the density of a liquid, it is a polynomial coefficient. It is not used for gases. **FLDATA8**,NOMI, **FLDATA9**,COF1, and **FLDATA10**,COF2, must also be used.

If the label is SPHT, the COF3 value is available in the user-programmable subroutine UserSpht.

See also the **FLDATA12**,PROP, and **FLDATA13**,VARY commands for other Fluid Property Definition commands.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties
Main Menu>Solution>FLOTRAN Set Up>Fluid Properties

FLDATA12, PROP, *Label*, *Value*

Sets the property update frequency flag.

PREP7: FLOTRAN Property

MP <> <> <> <> <> <> <> FL PP ED

PROP
Enter the word PROP in this field.

FLDATA12,PROP is the **FLDATA** command with its first argument set to PROP. It can be entered into the program as either **FLDATA12**,PROP,*Label*,*Value* or **FLDATA**,PROP,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Updating label (dictates the meaning of *Value*):

IVIS
Value is the initial guess for viscosity. If you do not specify any value, the nominal viscosity issued via the **FLDATA8**,NOMI,VISC command is the initial viscosity. For information on using this label, see the *ANSYS Fluids Analysis Guide*.

UFRQ
Value is the number of global iterations (frequency of updating) between the update of the properties.

Value
Number of iterations as described above (defaults to 1).

Notes

Properties will not be updated if the all the property types are set to CONSTANT with the **FLDATA7** command.

See also the **FLDATA7**,PROT, **FLDATA8**,NOMI, **FLDATA9**,COF1, **FLDATA10**,COF2, **FLDATA11**,COF3, and **FLDATA13**,VARY commands for other Fluid Property Definition commands.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties
Main Menu>Solution>FLOTRAN Set Up>Fluid Properties

FLDATA13, VARY, *Label*, *Value*
Sets the property variation flag.

PREP7: FLOTRAN Property
MP <> <> <> <> <> <> <> FL PP ED

VARY
Enter the word VARY in this field.

FLDATA13,VARY is the **FLDATA** command with its first argument set to VARY. It can be entered into the program as either **FLDATA13**,VARY,*Label*,*Value* or **FLDATA**,VARY,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Fluid property being described:

DENS
Density.

VISC
Viscosity.

COND
Thermal conductivity.

SPHT
Specific heat.

SFTS
Surface tension coefficient.

Value
Flag value for property variation:

T
To turn property variation on.

F
To turn property variation off (default).

Notes

For nonconstant fluid properties [**FLDATA7**,PROT], the appropriate flags must be set to T to allow property variation between global iterations.

See also the **FLDATA7**,PROT, **FLDATA8**,NOMI, **FLDATA9**,COF1, **FLDATA10**,COF2, **FLDATA11**,COF3, and **FLDATA12**,PROP commands for other Fluid Property Definition commands.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Fluid Properties
Main Menu>Solution>FLOTRAN Set Up>Fluid Properties

FLDATA14, TEMP, *Label*, *Value*

Specifies the reference temperature.

PREP7: FLOTRAN Operating

MP <> <> <> <> <> <> <> FL PP ED

TEMP
Enter the word TEMP in this field.

FLDATA14,TEMP is the **FLDATA** command with its first argument set to TEMP. It can be entered into the program as either **FLDATA14**,TEMP,*Label*,*Value* or **FLDATA**,TEMP,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Type of temperature specification (dictates the meaning of *Value*):

NOMI
Value is the initial temperature if a boundary condition or transient initial condition has not been set. *Value* defaults to 293.0.

BULK
Value is the temperature used to evaluate heat transfer coefficients given a heat flux or temperature at a boundary. *Value* defaults to 293.0.

TTOT
Value is the total (stagnation) temperature used in compressible adiabatic flow. The static temperature is calculated from the kinetic energy evaluated in terms of the velocity magnitude, specific heat, and gravitational constant:

$$T_{static} = T_{total} - \frac{V^2}{2 g_c C_p}$$

Value
Temperature as described above.

Notes

See also the **FLDATA15**,PRES, **FLDATA16**,BULK, and **FLDATA17**,GAMM commands for other Operating Condition commands.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Ref Conditions
Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Ref Conditions

---

FLDATA15, PRES, *Label*, *Value*

Specifies the reference pressure.

PREP7: FLOTRAN Operating

MP <> <> <> <> <> <> <> FL PP ED

PRES
Enter the word PRES in this field.

FLDATA15,PRES is the **FLDATA** command with its first argument set to PRES. It can be entered into the program as either **FLDATA15**,PRES,*Label*,*Value* or **FLDATA**,PRES,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Reference label (dictates the meaning of *Value*):

REFE
Value is the reference pressure. The absolute pressure results from adding the pressure components from rotating terms, the static pressure head, the FLOTRAN pressure, and the reference pressure. *Value* defaults to $1.0135 \times 10^{+5}$.

Value
Reference pressure as described above.

Notes

See also the **FLDATA14**,TEMP, **FLDATA16**,BULK, and **FLDATA17**,GAMM commands for other Operating Condition commands.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Ref Conditions
Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Ref Conditions

---

FLDATA16, BULK, *Label*, *Value*

Specifies the bulk modulus parameter.

PREP7: FLOTRAN Operating

MP <> <> <> <> <> <> <> FL PP ED

BULK
Enter the word BULK in this field.

FLDATA16,BULK is the **FLDATA** command with its first argument set to BULK. It can be entered into the program as either **FLDATA16**,BULK,*Label*,*Value* or **FLDATA**,BULK,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Label (dictates the meaning of *Value*):

BETA
Value is the bulk modulus parameter.

Value
Value of bulk modulus parameter (defaults to 10^{15}).

Notes

The bulk modulus parameter, β_p, is used in the transient algorithm for incompressible flows:

$$\beta_p = \frac{\partial p}{\partial p}$$

The default value corresponds to that of a constant density fluid.

See also the **FLDATA14**,TEMP, **FLDATA15**,PRES, and **FLDATA17**,GAMM commands for other Operating Condition commands.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Ref Conditions
Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Ref Conditions

---

FLDATA17, GAMM, *Label*, *Value*

Specifies the specific heat ratio.

PREP7: FLOTRAN Operating

MP <> <> <> <> <> <> <> FL PP ED

GAMM
Enter the word GAMM in this field.

FLDATA17,GAMM is the **FLDATA** command with its first argument set to GAMM. It can be entered into the program as either **FLDATA17**,GAMM,*Label*,*Value* or **FLDATA**,GAMM,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Label (dictates the meaning of *Value*):

COMP
Value is the ratio of specific heats.

Value
Value of the ratio (defaults to 1.4).

Notes

Specifies the ratio of specific heat at constant pressure to the specific heat at constant volume, Cp/Cv. It is used in compressible analyses.

See also the **FLDATA14**,TEMP, **FLDATA15**,PRES, and **FLDATA16**,BULK commands for other Operating Condition commands.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>Ref Conditions
Main Menu>Solution>FLOTRAN Set Up>Flow Environment>Ref Conditions

---

FLDATA18, METH, *Label*, *Value*

Selects the algebraic solver.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> <> FL PP ED

METH
Enter the word METH in this field.

FLDATA18,METH is the **FLDATA** command with its first argument set to METH. It can be entered into the program as either **FLDATA18**,METH,*Label*,*Value* or **FLDATA**,METH,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Degree of freedom set for which solver is being specified:

PRES
Pressure equation.

TEMP
Energy equation.

VX
U velocity.

VY
V velocity.

VZ
W velocity.

ENKE
Turbulent kinetic energy.

ENDS
Turbulent kinetic energy dissipation rate.

Value
Solver to be used for the degree of freedom set:

0
Do not solve the equation set.

1
Use the Tri-Diagonal Matrix Algorithm (TDMA) (default for all DOF sets except PRES and TEMP). Also see the **FLDATA19**,TDMA command.

2
Use the Conjugate Residual method. See also the **FLDATA20**,SRCH, **FLDATA21**,CONV, **FLDATA22**,MAXI, and **FLDATA23**,DELT commands.

3
Use the Preconditioned Conjugate Residual method (default for PRES DOF set). See also the **FLDATA20**,SRCH, **FLDATA21**,CONV, **FLDATA22**,MAXI, and **FLDATA23**,DELT commands.

4
Use the Preconditioned Generalized Minimum Residual (PGMR) solution method (default for TEMP DOF set). See FLOTRAN Thermal Analyses in the *ANSYS CFD Flotran Analysis Guide* for more information on conjugate heat transfer.

5
Use the Sparse Direct method.

6
Use the Preconditioned BiCGStab method (PBCGM).

Notes

A solver can be specified for each degree of freedom set. Repeat the **FLDATA18** command as needed.

The Tri-Diagonal Matrix Algorithm (TDMA) is a special case of the standard Gauss-Seidel iterative method for the solution of sets of algebraic equations. It is the preferred method for providing approximate solutions for the momentum and turbulence equations since exact solutions are not required. A convergence criterion is not specified for the TDMA method, merely the number of iterations (sweeps) to be performed [**FLDATA19**,TDMA].

Three methods are semi-direct solution methods based on search directions. The Conjugate Residual method requires the least memory, but stalls when solving ill-conditioned problems. (In an ill-conditioned problem, the thermal properties of fluid and non-fluid materials are different by several orders of magnitude). The Preconditioned Conjugate Residual method requires much more memory but performs better for ill-conditioned matrix problems which can arise when you are solving conjugate heat transfer problems. The Preconditioned Generalized Minimum Residual (PGMR) method is memory-intensive; by necessity, it incorporates a tight convergence criterion. The PGMR method is recommended for solving the energy equation for ill-conditioned conjugate transfer problems. In general, the Preconditioned BiCGStab method (PBCGM) requires less memory than the PGMR method. It is also recommended for extremely ill-conditioned conjugate heat transfer problems.

The Sparse Direct method is based on Gaussian elimination to factorize the matrix. This method is memory intensive and creates temporary files on the hard disk. It is robust and can be used for symmetric as well as non-symmetric equation systems.

For incompressible flow problems that involve the solution for the pressure degree of freedom, if you set *Value* equal to 2 or 3, the Preconditioned Conjugate Gradient method solver is used instead of the Conjugate Residual or Preconditioned Conjugate Residual method solvers.

The *ANSYS, Inc. Theory Reference* contains more detail on these methods.

See also the **FLDATA19**,TDMA, **FLDATA20**,SRCH, **FLDATA21**,CONV, **FLDATA22**,MAXI, and **FLDATA23**,DELT commands for other Algebraic Solver controls.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD

FLDATA19, TDMA, *Label*, *Value*

Specifies the number of TDMA sweeps.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> <> FL PP ED

TDMA
Enter the word TDMA in this field.

FLDATA19,TDMA is the **FLDATA** command with its first argument set to TDMA. It can be entered into the program as either **FLDATA19**,TDMA,*Label*,*Value* or **FLDATA**,TDMA,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Degree of freedom set for which the TDMA solver is being used:

PRES
Pressure equation (*Value* defaults to 100).

TEMP
Energy equation (*Value* defaults to 100).

VX
U velocity (*Value* defaults to 1).

VY
V velocity (*Value* defaults to 1).

VZ
W velocity (*Value* defaults to 1).

ENKE
Turbulent kinetic energy (*Value* defaults to 10).

ENDS
Turbulent kinetic energy dissipation rate (*Value* defaults to 10).

Value
Number of iterations (sweeps) for this degree of freedom set.

Notes

Specifies the number of iterations (sweeps) the Tri-Diagonal Matrix Algorithm [**FLDATA18**,METH] will perform during the solution. A different number of sweeps may be specified for each degree of freedom set the TDMA solver is used for. Increasing the number for the velocity DOF may cause instability.

See also the **FLDATA18**,METH command for other Algebraic Solver controls.

This command is accessible in the menu if **FLDATA18**,METH,*Label*,1 has been issued for the PRES, TEMP, ENKE, or ENDS labels.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD

FLDATA20, SRCH, *Label*, *Value*

Specifies the number of conjugate direction search vectors.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> <> FL PP ED

SRCH
Enter the word SRCH in this field.

FLDATA20,SRCH is the **FLDATA** command with its first argument set to SRCH. It can be entered into the program as either **FLDATA20**,SRCH,*Label*,*Value* or **FLDATA**,SRCH,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Degree of freedom set for which the solver is being used:

PRES
Pressure equation.

TEMP
Energy equation.

VX
U velocity.

VY
V velocity.

VZ
W velocity.

ENKE
Turbulent kinetic energy.

ENDS
Turbulent kinetic energy dissipation rate.

Value
Number of search directions (defaults to 2). If you are using the PGMR solver, the default is to use 12 search vectors. You cannot use fewer than 12 for the PGMR solver. If you are using the PBCGM solver, the number of search directions is 1 to 8, with 2 as the default.

Notes

The conjugate direction iterative techniques (methods 2 and 3 on the **FLDATA18**,METH command) develop a solution as a linear combination of search directions. In the solution with methods 2 and 3, new search vectors are made orthogonal to*Value* previous vectors in the solution of the non-symmetric matrix systems. See the *ANSYS Fluids Analysis Guide* for details on when to change these values.

Note — The parameter is not applicable to the incompressible pressure equation since the new search vector is automatically orthogonal to all the previous ones.

See also the **FLDATA18**,METH,TDMA, **FLDATA21**,CONV, **FLDATA22**,MAXI, and **FLDATA23**,DELT commands for other Algebraic Solver controls.

This command is accessible in the menu if **FLDATA18**,METH,*Label*,(2 or 3) has been issued for the relevant labels.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD

Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD

FLDATA20A, PGMR, *Label*, *Value*

Specifies the amount of fill-in when preconditioning the coefficient matrix.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> <> FL PP ED

PGMR

Enter the word PGMR in this field.

FLDATA20A,PGMR is the **FLDATA** command with its first argument set to PGMR. It can be entered into the program as either **FLDATA20A**,PGMR,*Label*,*Value* or **FLDATA**,PGMR,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label

Label (dictates the meaning of Value):

FILL

Value represents the number of extra elements allowed in each row of the L and U decomposition matrices. An extra element is defined as being in addition to the number of nonzero elements in the row of the original matrix. The allowable range for the fill parameter is 1 to 10 (defaults to 6).

MODP

Value represents the number of global iterations performed using the TDMA method between global iterations performed using the PGMR method for the temperature DOF. The first global iteration always uses the PGMR method. The global iteration count is based on the total number, not the number for a restart. Value defaults to 0 (PGMR always used).

Value

Value as described above.

Notes

The fill parameter specifies the amount of fill-in when constructing the L and U decomposition of the coefficient matrix.

Use of FILL requires selection of the PGMR solver for the PRES or TEMP degree of freedom. Use of MODP requires selection of the PGMR solver for the temperature DOF.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD

Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD

FLDATA20B, PBCGM, *Label*, *Value*

Specifies the number of fill-ins for the ILU preconditioner.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> <> FL PP ED

PBCGM
Enter the word PBCGM in this field.

FLDATA20B,PBCGM is the **FLDATA** command with its first argument set to PBCGM. It can be entered into the program as either **FLDATA20B**,PBCGM,*Label*,*Value* or **FLDATA**,PBCGM,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Label (dictates the meaning of Value):

FILL
Value represents the number of extra elements allowed in each row of the L and U decomposition matrices. An extra element is defined as being in addition to the number of nonzero elements in the row of the original matrix. The allowable range for the fill parameter is 0 to 10 (defaults to 6).

Value
Value as described above.

Notes

The fill parameter specifies the amount of fill-in when constructing the L and U decomposition of the coefficient matrix.

A single constant for FILL is used for the PBCGM solver, and it will be applied to all degrees of freedom that use this solver.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD

Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD

FLDATA21, CONV, *Label*, *Value*

Specifies the convergence criterion for FLOTRAN algebraic solvers.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> <> FL PP ED

CONV
Enter the word CONV in this field.

FLDATA21,CONV is the **FLDATA** command with its first argument set to CONV. It can be entered into the program as either **FLDATA21**,CONV,*Label*,*Value* or **FLDATA**,CONV,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Degree of freedom set for which the solver is being used:

PRES
Pressure equation (*Value* defaults to 1.0×10^{-12}).

TEMP
Energy equation (*Value* defaults to 1.0×10^{-12}).

VX
U velocity (*Value* defaults to 1.0×10^{-5}).

VY
V velocity (*Value* defaults to 1.0×10^{-5}).

VZ
W velocity (*Value* defaults to 1.0×10^{-5}).

ENKE
Turbulent kinetic energy (*Value* defaults to 1.0×10^{-5}).

ENDS
Turbulent kinetic energy dissipation rate (*Value* defaults to 1.0×10^{-5}).

Value
Convergence criterion factor.

Notes

The convergence monitor (for methods 2 or 3 on the **FLDATA18**,METH command) represents the factor by which the inner product of the residual vector is to be reduced during the solution of the equations at any global iteration.

If you are using the Preconditioned Generalized Minimum Residual (PGMR) solver, the least restrictive convergence criterion allowed is 1.E-10. If you try to use a less restrictive convergence criterion, FLOTRAN will change it to 1.E-10. A convergence criterion as low as 1.E-20 may be necessary for some problems.

Less restrictive values are specified for the velocities and turbulence parameters because the iterative nature of the segregated solution algorithm in FLOTRAN does not require exact solutions to these equations at any global iteration. The default solution method for these degrees of freedom is the Tri-Diagonal Matrix Algorithm (TDMA) method.

See also the **FLDATA18**,METH, **FLDATA20**,SRCH, **FLDATA22**,MAXI, and **FLDATA23**,DELT commands for other Algebraic Solver controls.

This command is accessible in the menu if **FLDATA18**,METH,*Label*,(2 or 3) has been issued for the relevant labels.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD

FLDATA22, MAXI, *Label*, *Value*

Specifies the maximum number of semi-direct iterations.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> <> FL PP ED

MAXI
Enter the word MAXI in this field.

FLDATA22,MAXI is the **FLDATA** command with its first argument set to MAXI. It can be entered into the program as either **FLDATA22**,MAXI,*Label*,*Value* or **FLDATA**,MAXI,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Degree of freedom set for which the solver is being used:

PRES
Pressure equation (*Value* defaults to 1000).

TEMP
Energy equation (*Value* defaults to 1000).

VX
U velocity (*Value* defaults to 100).

VY
V velocity (*Value* defaults to 100).

VZ
W velocity (*Value* defaults to 100).

ENKE
Turbulent kinetic energy (*Value* defaults to 100).

ENDS
Turbulent kinetic energy dissipation (*Value* defaults to 100).

Value
Limit on the number of iterations.

Notes

These limits apply to the semi-direct solution methods (method 2 or 3 on the **FLDATA18**,METH command). If this number of iterations is reached before the convergence criterion is met, the solution is accepted, a warning message is printed, and the program continues normally. However if the pressure equation is not solved to the precision desired three times during a load step, execution will terminate.

See also the **FLDATA18**,METH, **FLDATA20**,SRCH, **FLDATA21**,CONV, and **FLDATA23**,DELT commands for other Algebraic Solver controls.

This command is accessible in the menu if **FLDATA18**,METH,*Label*,(2 or 3) has been issued for the relevant labels.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD

FLDATA23, DELT, *Label*, *Value*

Specifies the solver minimum normalized rate of change.

PREP7: FLOTRAN Solver

MP <> <> <> <> <> <> <> FL PP ED

DELT

Enter the word DELT in this field.

FLDATA23,DELT is the **FLDATA** command with its first argument set to DELT. It can be entered into the program as either **FLDATA23**,DELT,*Label*,*Value* or **FLDATA**,DELT,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label

Degree of freedom set for which the solver is being used:

PRES

Pressure equation (*Value* defaults to 1.0 x 10^{-10}).

TEMP

Energy equation (*Value* defaults to 1.0 x 10^{-10}).

VX

U velocity (*Value* defaults to 1.0 x 10^{-10}).

VY

V velocity (*Value* defaults to 1.0 x 10^{-10}).

VZ

W velocity (*Value* defaults to 1.0 x 10^{-10}).

ENKE

Turbulent kinetic energy (*Value* defaults to 1.0 x 10^{-10}).

ENDS

Turbulent kinetic energy dissipation (*Value* defaults to 1.0 x 10^{-10}).

Value

Minimum normalized rate of change (delta).

Notes

Delta is the minimum normalized rate of change which will permit the semi-direct solution methods (method 2 or 3 on the **FLDATA18**,METH command) to continue.

Delta is used to terminate the semi-direct solvers in the event that stall occurs. If the methods stall, the solver increments the solution only a very small amount despite the fact that the correct solution has not been achieved. The maximum nodal difference between the solutions, normalized to the value of the variable, is compared to delta.

Termination of the algebraic solver due to the small rate of change is considered a normal function and no warning message is printed. Execution of FLOTRAN continues normally.

See also the **FLDATA18**,METH, **FLDATA20**,SRCH, **FLDATA21**,CONV, and **FLDATA22**,MAXI commands for other Algebraic Solver controls.

This command is accessible in the menu if **FLDATA18**,METH,*Label*,(2 or 3) has been issued for the relevant labels.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Preprocessor>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENDS Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>ENKE Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>PRES Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>TEMP Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VX Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VY Solver CFD
Main Menu>Solution>FLOTRAN Set Up>CFD Solver Controls>VZ Solver CFD

FLDATA24, TURB, *Label*, *Value*

Sets the turbulence model and the constants used in the Standard k-ε Model and the Zero Equation Turbulence Model.

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> <> FL PP ED

TURB
Enter the word TURB in this field.

FLDATA24,TURB is the **FLDATA** command with its first argument set to TURB. It can be entered into the program as either **FLDATA24**,TURB,*Label*,*Value* or **FLDATA**,TURB,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Turbulence model parameter label (dictates the meaning of *Value*):

MODL
Value sets the turbulence model used. Valid values are the integers 0 through 8 as follows:

0,1
Standard k-ε Model (default).

2
Zero Equation Turbulence Model.

3
Re-Normalized Group Turbulence Model (RNG).

4
New k-ε Model due to Shih (NKE).

5
Nonlinear Model of Girimaji (GIR).

6
Shih, Zhu, Lumley Model (SZL).

7
k-ω Turbulence Model

8
Shear-Stress Transport Turbulence Model (SST)

RATI
Value is the turbulence ratio. The initial turbulent viscosity is equal to the laminar value times the turbulence ratio. *Value* defaults to 1000.

ININ
Value is the inlet intensity. The fluctuating velocity component at the inlet is assumed to be the magnitude of the inlet velocity times the inlet intensity. *Value* defaults to 0.01.

INSF
Value is the inlet scale factor. This factor is used to provide a length scale inlet region of the problem. The hydraulic diameter of the inlet is multiplied by the inlet scale factor. If a negative value is input, the absolute value is used as direct input of the hydraulic diameter. *Value* defaults to 0.01.

SCTK
Value is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 1.0.

SCTD
Value is the Schmidt number for the kinetic energy dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 1.3.

SCTM
Value is the Schmidt number for the momentum equation. The turbulent contribution to the diffusion term of the momentum equation is divided by this factor. *Value* defaults to 1.

SCTT
Value is the Schmidt number for the energy (temperature) equation. The turbulent contribution to the diffusion term of the energy equation is divided by this factor. The Prandtl number is equal to the Schmidt number. *Value* defaults to 0.85.

CMU
Value is the k-ε turbulence model constant

$$\mu_t = C_\mu \frac{\rho k^2}{\varepsilon}$$

which is used in the update of the turbulent viscosity. *Value* defaults to 0.09.

C1
Value is the k-ε turbulence model C1 constant. It is the multiplier of the shear rate generation term of the turbulent kinetic energy dissipation rate equation. *Value* defaults to 1.44.

C2
Value is the k-ε turbulence model C2 constant. It is the multiplier of the dissipation source term in the turbulent kinetic energy dissipation rate equation. *Value* defaults to 1.92.

BUC3
Value is the k-ε buoyancy model constant. A value of zero means that there is no contribution to the turbulent kinetic energy dissipation rate equation. The default value of one is appropriate for stable thermally stratified flows. A value of zero is appropriate for unstable thermally stratified flows

BUC4

Value is the k-ε multiplier applied to the buoyancy term of the turbulent kinetic energy equation. A value of 1.0 is appropriate for the calculation of stable thermally stratified flows. *Value* defaults to 0.0.

BETA

Value is β, the coefficient of thermal expansion

$$\beta = \frac{1}{\rho}\frac{\partial \rho}{\partial T}$$

This term is used in the buoyancy terms of the k-ε model.

KAPP

Value is the slope parameter of the law of the wall constant. It is the slope of the plot of normalized shear velocity (u^+) versus the nondimensionalized distance from the wall (y^+). See *ANSYS, Inc. Theory Reference* for details. *Value* defaults to 0.4.

EWLL

Value is the law of the wall constant. It is related to the y intercept value for a plot of normalized shear velocity (u^+) versus the nondimensionalized distance from the wall (y^+). See the *ANSYS, Inc. Theory Reference* for more details. *Value* defaults to 9.0.

WALL

Value is the choice of wall conductivity model. The default model is the Van Driest model (*Value* = VAND), used most often for high Prandtl number fluids. The second choice is the Spalding model (*Value* = SPAL), applicable to low Prandtl number fluids. The third choice is the Equilibrium model (*Value* = EQLB). The equilibrium model is also automatically invoked for the wall viscosity by this command.

VAND

Value is the constant in the Van Driest wall conductivity model. See the *ANSYS, Inc. Theory Reference* for details. *Value* defaults to 26.0.

TRAN

Value is the magnitude of y^+ marking the outer boundary of the laminar sublayer. Used only for the Equilibrium Wall model. Value defaults to 11.5.

ZELS

Value is the Zero Equation Model length scale (defaults to -1). A negative value means that FLOTRAN will calculate the value internally.

KS

Value is the local uniform wall roughness in length units. The default value of 0.0 implies a smooth wall.

CKS

Value is an empirical dimensionless factor between 0.5 and 1.0 that specifies the degree of nonuniformity of the surface. The default value of 0.5 means that the roughness signified by KS is uniformly distributed. Higher values increase the roughness losses without changing the flow regime implied by the value of KS.

Value

Turbulence model parameters values as explained above.

Notes

Sets the turbulence model and constants used in the Standard k-ε Model and the Zero Equation Turbulence Model. The other turbulence models are an extension of the Standard k-ε Model and different values are applied to some of the Standard k-ε Model constants and some additional constants are added.

Applicable only if the turbulence model is activated [**FLDATA1**,SOLU,TURB, TRUE]. The default values are used most often. See the *ANSYS, Inc. Theory Reference* for more information on the turbulence models.

See also the **FLDATA24A**,RNGT, **FLDATA24B**,NKET, **FLDATA24C**,GIRT, and **FLDATA24D**,SZLT commands for other turbulence model constants.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Buoyancy Terms
Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model
Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Parameters
Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Wall Parameters
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Buoyancy Terms
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Parameters
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Wall Parameters

FLDATA24A, RNGT, *Label*, *Value*

Sets constants for the Re-Normalized Group Turbulence Model (RNG).

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> <> FL PP ED

RNGT
Enter the word RNGT in this field.

FLDATA24A,RNGT is the **FLDATA** command with its first argument set to RNGT. It can be entered into the program as either **FLDATA24A**, RNGT, *Label*,*Value* or **FLDATA**,RNGT,*Label*,*Value* where *Label* and *Value* are as described below.

Label
Turbulence model parameter label (dictates the meaning of *Value*):

SCTK
Value is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 0.72.

SCTD
Value is the Schmidt number for the kinetic energy dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 0.72.

CMU
Value is the turbulence model constant

$$\mu_t = C_\mu \frac{\rho k^2}{\varepsilon}$$

which is used in the update of the turbulent viscosity. *Value* defaults to 0.085.

C1
Value is the multiplier of the shear rate generation term of the turbulent kinetic energy dissipation rate equation. *Value* defaults to 1.42.

C2
Value is the multiplier of the dissipation source term in the turbulent kinetic energy dissipation rate equation. *Value* defaults to 1.68.

BETA
Value is the RNG model constant,
β_∞

Value defaults to 0.012, which corresponds to a value of 0.4 for the von Karman constant.

ETAI
Value is the asymptotic value of the strain rate parameter eta. *Value* defaults to 4.38.

Value
Turbulence model parameters values as explained above.

Notes

The Re-Normalized Group Turbulence Model (RNG) is an extension of the Standard k-ε Model. Different values are applied to five of the Standard k-ε Model constants and two additional constants are added. The values for the RNG Model are applied with the **FLDATA24A**,RNGT command and are separate from the Standard k-ε Model constants.

Applicable only if the Re-Normalized Group Turbulence Model (RNG) is activated [**FLDATA24**,TURB,MODL,3]. The default values are used most often. See the *ANSYS Theory Reference* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

---

FLDATA24B, NKET, *Label*, *Value*

Sets constants for the k-ε Turbulence Model due to Shih (NKE).

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> <> FL PP ED

NKET
Enter the word NKET in this field.

FLDATA24B,NKET is the **FLDATA** command with its first argument set to NKET. It can be entered into the program as either **FLDATA24B**,NKET, *Label*,*Value* or **FLDATA**,NKET,*Label*,*Value* where *Label* and *Value* are as described below.

Label
Turbulence model parameter label (dictates the meaning of *Value*):

SCTK
Value is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. It is the σ_k constant. *Value* defaults to 1.0.

SCTD
Value is the Schmidt number for the kinetic energy dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. It is the σ_ε constant. *Value* defaults to 1.2.

C2
Value is the multiplier of the dissipation source term in the turbulent kinetic energy dissipation rate equation. *Value* defaults to 1.90.

C1MX
Value is the maximum allowed value of the C1 constant in the turbulent kinetic energy dissipation rate equation. *Value* defaults to 0.43.

Value
Turbulence model parameters values as explained above.

Notes

The k-ε Turbulence Model due to Shih (NKE) is an extension of the Standard k-ε Model. Different values are applied to three of the Standard k-ε Model constants and an additional constant is added. The values for the NKE Model are applied with the **FLDATA24B**,NKET command and are separate from the Standard k-ε Model constants.

Applicable only if the k-ε Model due to Shih (NKE) is activated [**FLDATA24**,TURB,MODL,4]. The default values are used most often. See the *ANSYS, Inc. Theory Reference* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

---

FLDATA24C, GIRT, *Label*, *Value*

Sets constants for the Nonlinear Turbulence Model of Girimaji (GIR).

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> <> FL PP ED

GIRT
Enter the word GIRT in this field.

FLDATA24C,GIRT is the **FLDATA** command with its first argument set to GIRT. It can be entered into the program as either **FLDATA24C**,GIRT,*Label*,*Value* or **FLDATA**,GIRT,*Label*,*Value* where *Label* and *Value* are as described below.

Label
Turbulence model parameter label (dictates the meaning of *Value*):

SCTK
Value is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 1.0.

SCTD
Value is the Schmidt number for the kinetic energy dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 1.3.

G0
Value is the C_1^0 constant. *Value* defaults to 3.6.

G1
Value is the C_1^1 constant. *Value* defaults to 0.0.

G2
Value is the C_2 constant. *Value* defaults to 0.8.

G3
Value is the C_3 constant. *Value* defaults to 1.94.

G4
Value is the C_4 constant. *Value* defaults to 1.16.

Value
Turbulence model parameters values as explained above.

Notes

Sets constants for the Nonlinear Turbulence Model of Girimaji (GIR), which is an extension of the Standard k-ε Model. Different values are applied to two of the Standard k-ε Model constants and five additional constants are added. The values for the GIR Model are applied with the **FLDATA24C**,GIRT command and are separate from the Standard k-ε Model constants.

Applicable only if the Nonlinear Model of Girimaji (GIR) is activated [**FLDATA24**,TURB,MODL,5]. The default values are used most often. See the *ANSYS, Inc. Theory Reference* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

---

FLDATA24D, SZLT, *Label*, *Value*

Sets constants for the Shih, Zhu, Lumley Turbulence Model (SZL).

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> <> FL PP ED

SZLT
Enter the word SZLT in this field.

FLDATA24D,SZLT is the **FLDATA** command with its first argument set to SZLT. It can be entered into the program as either **FLDATA24D**,SZLT,*Label*,*Value* or **FLDATA**,SZLT,*Label*,*Value* where *Label* and *Value* are as described below.

Label

Turbulence model parameter label (dictates the meaning of Value):

SCTK

Value is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 1.0.

SCTD

Value is the Schmidt number for the kinetic energy dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 1.3.

SZL1

Value is the numerator constant used in the calculation of C_μ. It is the A_{szl1} constant. *Value* defaults to 0.666.

SZL2

Value is the denominator constant used in the calculation of C_μ. It is the A_{szl2} constant. *Value* defaults to 1.25.

SZL3

Value is the strain rate multiplier. It is the A_{szl3} constant. Value defaults to 0.90.

Value

Turbulence model parameters values as explained above.

Notes

Sets constants for the Shih, Zhu, Lumley Turbulence Model (SZL), which is an extension of the Standard k-ε Model. Different values are applied to two of the Standard k-ε Model constants and three additional constants are added. The values for the SZL Model are applied with the **FLDATA24D**,SZLT command and are separate from the Standard k-ε Model constants.

Applicable only if the Shih, Zhu, Lumley Model (SZL) is activated [**FLDATA24**,TURB,MODL,6]. The default values are used most often. See the *ANSYS, Inc. Theory Reference* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

FLDATA24E, SKWT, *Label*, *Value*

Sets constants for the k-ω turbulence model.

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> <> FL PP ED

SKWT
Enter the word SKWT in this field.

FLDATA24E,SKWT is the **FLDATA** command with its first argument set to SKWT. It can be entered into the program as either **FLDATA24E**,SKWT,*Label*,*Value* or **FLDATA**,SKWT,*Label*,*Value* where *Label* and *Value* are as described below.

Label
Turbulence model parameter label (dictates the meaning of *Value*):

SCTK
Value is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 2.0.

SCTW
Value is the Schmidt number for the specific dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 2.0.

GAMA
Value is the GAMMA factor. *Value* defaults to 0.5555.

BETA
Value is the BETA factor. *Value* defaults to 0.075.

Value
Turbulence model parameters values as explained above.

Notes

Sets constants for the k-ω Turbulence Model. Values for the k-ω model are applied with the **FLDATA24E**,SKWT command.

Applicable only if the k-ω model is activated [**FLDATA24**,TURB,MODL,7]. The default values are used most often. See the *ANSYS, Inc. Theory Reference* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

FLDATA24F, SST1, *Label*, *Value*

Sets the turbulent production clip factor for the Shear Stress Transport (SST) turbulence model.

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> <> FL PP ED

SST1
Enter the word SST1 in this field.

FLDATA24F,SST1 is the **FLDATA** command with its first argument set to SST1. It can be entered into the program as either **FLDATA24F**,SST1,*Label*,*Value* or **FLDATA**,SST1,*Label*,*Value* where *Label* and *Value* are as described below.

Label
Enter the word CLMT in this field.

Value
Turbulent production clip factor. Defaults to 1.0e15 (no clipping).

Notes

Sets the turbulent production clip factor for the Shear Stress Transport (SST) turbulence model. Values for the k-ω regime are applied with the **FLDATA24F**,SST1,CLMT command.

Applicable only if the Shear Stress Transport model is activated [**FLDATA24**,TURB,MODL,8]. The default values are used most often. See the *ANSYS, Inc. Theory Reference* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

---

FLDATA24G, SST1, *Label*, *Value*

Sets constants in the k-ω regime for the Shear Stress Transport (SST) turbulence model.

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> <> FL PP ED

SST1
Enter the word SST1 in this field.

FLDATA24G,SST1 is the **FLDATA** command with its first argument set to SST1. It can be entered into the program as either **FLDATA24G**,SST1,*Label*,*Value* or **FLDATA**,SST1,*Label*,*Value* where *Label* and *Value* are as described below.

Label
Turbulence model parameter label (dictates the meaning of *Value*):

SCTK
Value is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 1.176.

SCTW
Value is the Schmidt number for the specific dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 2.0.

GAMA
Value is the GAMMA factor.*Value* defaults to 0.5532.

BETA
Value is the BETA factor. *Value* defaults to 0.075.

Value
Turbulence model parameter values as explained above.

Notes

Sets constants in the k-ω regime for the Shear Stress Transport (SST) turbulence model. Values for the k-ω regime are applied with the **FLDATA24G**,SST1 command.

Applicable only if the Shear Stress Transport model is activated [**FLDATA24**,TURB,MODL,8]. The default values are used most often. See the *ANSYS, Inc. Theory Reference* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

---

FLDATA24H, SST2, *Label*, *Value*

Sets constants in the k-ε regime for the Shear Stress Transport (SST) turbulence model.

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> <> FL PP ED

SST2
Enter the word SST2 in this field.

FLDATA24H,SST2 is the **FLDATA** command with its first argument set to SST2. It can be entered into the program as either **FLDATA24H**,SST2,*Label*,*Value* or **FLDATA**,SST2,*Label*,*Value* where *Label* and *Value* are as described below.

Label
Turbulence model parameter label (dictates the meaning of *Value*):

SCTK
Value is the Schmidt number for the turbulent kinetic energy. The diffusion term in the turbulent kinetic energy equation is divided by this factor. *Value* defaults to 1.0.

SCTW
Value is the Schmidt number for the specific dissipation rate. The diffusion term in the dissipation rate equation is divided by this factor. *Value* defaults to 1.168.

GAMA
Value is the GAMMA factor. *Value* defaults to 0.4403.

BETA
Value is the BETA factor. *Value* defaults to 0.0828.

Value
Turbulence model parameter values as explained above.

Notes

Sets constants in the k-ε regime for the Shear Stress Transport (SST) turbulence model. Values for the k-ε regime are applied with the **FLDATA24H**,SST2 command.

Applicable only if the Shear Stress Transport model is activated [**FLDATA24**,TURB,MODL,8]. The default values are used most often. See the *ANSYS, Inc. Theory Reference* for more information on the turbulence models.

See also the **FLDATA24**,TURB command.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Turbulence>Turbulence Model
Main Menu>Solution>FLOTRAN Set Up>Turbulence>Turbulence Model

---

FLDATA25, RELX, *Label*, *Value*

Sets solution and property relaxation factors.

PREP7: FLOTRAN Stability

MP <> <> <> <> <> <> <> FL PP ED

RELX
Enter the word RELX in this field.

FLDATA25,RELX is the **FLDATA** command with its first argument set to RELX. It can be entered into the program as either **FLDATA25**,RELX,*Label*,*Value* or **FLDATA**,RELX,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Solution and property relaxation factor labels:

VX
Value is the U Velocity relaxation factor.

VY
Value is the V Velocity relaxation factor.

VZ
Value is the W Velocity relaxation factor.

PRES
Value is the pressure relaxation factor.

TEMP
Value is the temperature relaxation factor.

ENKE
Value is the kinetic energy relaxation factor.

ENDS
Value is the dissipation rate relaxation factor.

EVIS
Value is the effective viscosity relaxation factor.

ECON
Value is the effective conductivity relaxation factor.

DENS
Value is the density update relaxation factor.

VISC
Value is the viscosity update relaxation factor.

COND
Value is the conductivity update relaxation factor.

SPHT
Value is the specific heat relaxation factor.

Value
Value of relaxation parameter as explained above (defaults to 0.5). Defaults to 1.0 for SPHT.

Notes

For stability purposes the solution in FLOTRAN is under-relaxed between global iterations. When the solution of the equations for a given degree of freedom is completed, the actual set of values used is related to the calculated set and the previous values as follows (where RELX is the relaxation factor):

$$\Phi_{new} = (1 - RELX)\ \Phi_{previous} + RELX\ \Phi_{calculated}$$

See also the **FLDATA18**,METH, **FLDATA19**,TDMA, **FLDATA20**,SRCH, **FLDATA21**,CONV, **FLDATA22**,MAXI, and **FLDATA23**,DELT commands for other Solver Stability controls.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>DOF Relaxation
Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>Prop Relaxation
Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>DOF Relaxation
Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>Prop Relaxation

---

FLDATA26, STAB, *Label*, *Value*

Sets stability controls.

PREP7: FLOTRAN Stability

MP <> <> <> <> <> <> <> FL PP ED

STAB
Enter the word STAB in this field.

FLDATA26,STAB is the **FLDATA** command with its first argument set to STAB. It can be entered into the program as either **FLDATA26**,STAB,*Label*,*Value* or **FLDATA**,STAB,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Control label (dictates the meaning of *Value*):

TURB
Turbulence inertial relaxation (*Value* defaults to 1.0 x 10^{15}).

MOME
Momentum inertial relaxation (*Value* defaults to 1.0 x 10^{15}).

PRES
Pressure inertial relaxation (*Value* defaults to 1.0 x 10^{15}).

TEMP
Energy inertial relaxation (*Value* defaults to 1.0 x 10^{20}).

VISC
Artificial viscosity (*Value* defaults to 0.0).

Value
Inertial relaxation factor.

Notes

These controls are used to make the sets of equations diagonally dominant through inertial relaxation. Making the matrix equations more diagonally dominant makes them easier to solve. More global iterations are required for convergence if inertial relaxation is used. See the *ANSYS, Inc. Theory Reference* for details.

Artificial viscosity can be activated to enhance stability in difficult compressible problems as well as in some incompressible problems. The artificial viscosity is added to the main diagonal and the forcing function of the momentum equations. The equations become more diagonally dominant locally in regions with high velocity gradients.

In compressible analyses, the artificial viscosity should be gradually removed as convergence is achieved since a nonzero value will affect the final solution. This is not necessary for incompressible analyses as the divergence of velocity should be zero. In practice, values of artificial viscosity should not exceed 1000 times the effective viscosity.

See also the **FLDATA18**,METH, **FLDATA19**,TDMA, **FLDATA20**,SRCH, **FLDATA21**,CONV, **FLDATA22**,MAXI, **FLDATA23**,DELT, **FLDATA24**,TURB, and **FLDATA25**,RELX commands for other Solver Stability controls.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>Stability Parms
Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>Stability Parms

FLDATA27, PRIN, *Label*, *Value*

Controls dependent variable printing.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

PRIN
Enter the word PRIN in this field.

FLDATA27,PRIN is the **FLDATA** command with its first argument set to PRIN. It can be entered into the program as either **FLDATA27**,PRIN,*Label*,*Value* or **FLDATA**,PRIN,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Dependent variable tabulation is being specified for:

VX
U velocity.

VY
V velocity.

VZ
W velocity.

PRES
Pressure.

TEMP
Temperature.

ENKE
Kinetic energy.

ENDS
Turbulent kinetic energy dissipation rate.

DENS
Density.

VISC
Viscosity.

COND
Conductivity.

SPHT
Specific heat.

EVIS
Effective viscosity.

ECON
Effective conductivity.

SFTS
Surface tension coefficient.

Value
Tabulation key:

T
Turn tabulation on.

F
Turn tabulation off (default).

Notes

When set to T, this control produces a tabulation of the values of the particular dependent variable at every node. It is set at the beginning of a load step. The printout can be large and is contained in the file **Jobname.PFL**.

To observe the values of a dependent variable at selected nodes during postprocessing, see the **PRNSOL** command.

Menu Paths

This command cannot be accessed from a menu.

FLDATA28, MODR, *Label*, *Value*

Specifies that variable results are to be replaced.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

MODR
Enter the word MODR in this field.

FLDATA28,MODR is the **FLDATA** command with its first argument set to MODR. It can be entered into the program as either **FLDATA28**,MODR,*Label*,*Value* or **FLDATA**,MODR,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Dependent variables to be replaced:

VX
U velocity.

VY
V velocity.

VZ
W velocity.

PRES
Pressure.

TEMP
Temperature.

ENKE
Kinetic energy.

ENDS
Turbulent kinetic energy dissipation rate.

DENS
Density.

VISC
Viscosity.

COND
Conductivity.

SPHT
Specific heat.

EVIS
Effective viscosity.

ECON
Effective conductivity.

TTOT
Total temperature.

SFTS
Surface tension coefficient.

Value
Variable replace flag:

T
Allow this variable to be re-initialized.

F
Do not allow this variable to be re-initialized (default).

Notes

Results for any of the dependent variables available can be replaced with new values for the next load step. Only a single value may be specified for the entire solution domain. This is used for re-initialization of a variable such as temperature, which has diverged while other temperature independent calculations have been successful.

To re-initialize a variable, first use this command to set the replacement flag to T for the desired variable. Then use the **FLDATA29**,MODV command to put in the new variable value. Note that the flag(s) will be automatically reset to F after the new value has been inserted.

See also the **FLDATA29**,MODV command.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Mod Res/Quad Ord>Modify Results
Main Menu>Solution>FLOTRAN Set Up>Mod Res/Quad Ord>Modify Results

FLDATA29, MODV, *Label*, *Value*

Re-initializes a results variable.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

MODV
Enter the word MODV in this field.

FLDATA29,MODV is the **FLDATA** command with its first argument set to MODV. It can be entered into the program as either **FLDATA29**,MODV,*Label*,*Value* or **FLDATA**,MODV,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Dependent variables to be replaced:

VX
U velocity.

VY
V velocity.

VZ
W velocity.

SP0*n*
Mass fraction of species *n*, where *n* = 1 to 6 (FLOTRAN). If a species is given a user-defined name [**MSSPEC**], use that name instead of SP0*n*.

PRES
Pressure.

TEMP
Temperature.

ENKE
Kinetic energy.

ENDS
Turbulent kinetic energy dissipation rate.

DENS
Density.

VISC
Viscosity.

COND
Conductivity.

SPHT
Specific heat.

EVIS
Effective viscosity.

ECON
Effective conductivity.

TTOT
Total temperature.

SFTS
Surface tension coefficient.

LMD*n*
Laminar mass diffusion coefficient for species *n*, where *n* = 1 to 6.

EMD*m*
Effective mass diffusion coefficient for species *n*, where n = 1 to 6.

Value
New value of variable (defaults to 0.0). Specifying **FLDATA29**,MODV,*Label*,*Value* modifies the entire field variable to a constant value.

Notes

Results for any of the dependent variables available can be replaced with new values for the next load step. Only a single value may be specified for the entire solution domain. This is used for re-initialization of a variable such as temperature, which has diverged while other temperature independent calculations have been successful. Boundary conditions are reapplied upon restart.

Quantities which are not allowed to vary (e.g., CONSTANT properties) should not be modified. Also, modifying results should not be attempted on the initial run.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Mod Res/Quad Ord>Modify Results
Main Menu>Solution>FLOTRAN Set Up>Mod Res/Quad Ord>Modify Results

---

FLDATA30, QUAD, *Label*, *Value*

Controls the quadrature orders.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

QUAD
Enter the word QUAD in this field.

FLDATA30,QUAD is the **FLDATA** command with its first argument set to QUAD. It can be entered into the program as either **FLDATA30**,QUAD,*Label*,*Value* or **FLDATA**,QUAD,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Element integral to change quadrature for:

MOMD
Momentum diffusion term (defaults to 0 integration points).

MOMS
Momentum source term (defaults to 0 integration points).

PRSD
Pressure diffusion term (defaults to 1 integration point).

PRSS
Pressure source term (defaults to 1 integration point).

THRD
Thermal diffusion term (defaults to 0 integration points).

THRS
Thermal source term (defaults to 0 integration points).

TRBD
Turbulent diffusion terms (defaults to 0 integration points).

TRBS
Turbulent source terms (defaults to 2 integration points).

Value
Number of integration points.

Notes

Controls the number of integration points used in the evaluation of element integrals. They are set at the optimum values by default. Values of 0 and 1 correspond to 1 point quadrature, but 0 means an average value of the diffusion coefficient has been used in the integrals. Values are automatically set to 2 for axisymmetric and polar analyses. Using a value of 2 will improve the accuracy for analyses using distorted elements. Repeat the **FLDATA30** command as required.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Mod Res/Quad Ord>CFD Quad Orders
Main Menu>Solution>FLOTRAN Set Up>Mod Res/Quad Ord>CFD Quad Orders

---

FLDATA31, CAPP, *Label*, *Value*

Specifies dependent variable caps.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

CAPP
Enter the word CAPP in this field.

FLDATA31,CAPP is the **FLDATA** command with its first argument set to CAPP. It can be entered into the program as either **FLDATA31**,CAPP,*Label*,*Value* or **FLDATA**,CAPP,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Capping parameters. The first three are used to set the flags, and the rest are used to specify the cap values after the flag is set to T:

VELO
Value is the velocity capping flag (*Value* defaults to F).

TEMP
Value is the temperature capping flag (*Value* defaults to F).

PRES
Value is the pressure capping flag (*Value* defaults to F).

UMIN
Value is the minimum value of U velocity allowed (*Value* defaults to -1.E20).

UMAX
Value is the maximum value of U velocity allowed (*Value* defaults to +1.E20).

VMIN
Value is the minimum value of V velocity allowed (*Value* defaults to -1.E20).

VMAX
Value is the maximum value of V velocity allowed (*Value* defaults to +1.E20).

WMIN
Value is the minimum value of W velocity allowed (*Value* defaults to -1.E20).

WMAX
Value is the maximum value of W velocity allowed (*Value* defaults to +1.E20).

TMIN
Value is the minimum value of temperature allowed (*Value* defaults to 0.0).

TMAX
Value is the maximum value of temperature allowed (*Value* defaults to +1.E20).

PMIN
Value is the minimum value of pressure allowed (*Value* defaults to -1.E20).

PMAX
Value is the maximum value of pressure allowed (*Value* defaults to +1.E20).

Value
Capping flag (T or F) or capping parameter value.

Notes

These parameters are used to limit arbitrarily the values of the dependent variables. Capping helps prevent divergence in the early stages of analyses. Be careful when using caps to ensure that they have no impact on the final answers. You should remove capping as convergence is approached.

To use capping, you must first set the flag to T and then set the maximum and minimum caps.

The pressure value calculated by the solution of the pressure equation is capped, not the relaxed value. Therefore, if you introduce pressure capping upon restarting an analysis, pressure values may still be outside the caps.

Capping applies to relative values of pressure and absolute values of temperature.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>Results Capping
Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>Results Capping

FLDATA32, REST, *Label*, *Value*, *Value2*, *Fname*, *Ext*, --

Controls restart options.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

REST

Enter the word REST in this field.

FLDATA32,REST is the **FLDATA** command with its first argument set to REST. It can be entered into the program as either **FLDATA32**,REST,*Label*,*Value*,*Value2* or **FLDATA**,REST,*Label*,*Value*,*Value2* where *Label*, *Value*, and *Value2* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label

Restart option (dictates the meaning of *Value* and *Value2*):

NSET

Identifies the restart point by results set number. *Value* is the results set number in the results file (**Jobname.RFL**) from which the analysis will be restarted. *Value* = 0 or blank indicates that the restart will be from the last results set. *Value* will be reset to zero if the label ITER, LSTP, or TIME is subsequently set. *Value2* is not used.

ITER

Identifies the restart point by cumulative iteration number. *Value* is the cumulative iteration number in the results file (**Jobname.RFL**) from which the analysis will be restarted. If there are no results stored for this cumulative iteration, the results set with the next lowest cumulative iteration number will be used. *Value* = 0 or blank indicates that the restart will be from the last results set. *Value* will be reset to zero if the label NSET, LSTP, or TIME is subsequently set. *Value2* is not used.

LSTP

Identifies the restart point by load step and substep numbers. *Value* and *Value2* are the load step number and substep number in the results file (**Jobname.RFL**) from which the analysis will be restarted. *Value* = 0 or blank indicates that the restart will be from the last results set. *Value* and *Value2* will be reset to zero if the label ITER, NSET, or TIME is subsequently set.

TIME

Identifies the restart point by transient time. *Value* is the time point in the results file (**Jobname.RFL**) from which the analysis will be restarted. If there are no results stored for this time point, the results set with the next lowest time point will be used. *Value* = 0 or blank indicates that the restart will be from the last results set. *Value* will be reset to zero if the label NSET, LSTP, or ITER is subsequently set. *Value2* is not used.

RFIL

Specifies whether the CFD data structure restart file (**Jobname.CFD**) is to be read for the restart. Useful for large models where the creation of the data structures may take a long time. *Value* may be T (true) or F (false) and defaults to F. If the **Jobname.CFD** file does not exist, it will be created (if RFIL is set to true). Setting RFIL to true will toggle WFIL to false, and setting WFIL to true will toggle RFIL to false.

WFIL

Specifies whether the CFD data structure restart file (**Jobname.CFD**) is to be written. Useful for overwriting an existing restart file when changes in the model or boundary conditions have occurred. *Value* may be T (true) or F (false) and defaults to F. Setting WFIL to true will toggle RFIL to false, and setting RFIL to true will toggle WFIL to false.

OVER
Specifies whether to overwrite the set of results from which the restart occurs. *Value* may be -1, 0, or 1 (defaults to 0). If *Value* is -1, the previous set of results are overwritten. If *Value* is 1, the previous set of results is saved. If *Value* is 0 (default), the previous results are saved only if the results were written as a saved (converged) set of results. When this flag is used to change the status of the previous set of results, ANSYS sets it to 0 so that future sets of results are not affected.

CLEAR
Specifies whether to eliminate from the results file (**Jobname.RFL**) all results sets stored before and after the set used for the restart. Value may be T (true) or F (false) and defaults to F. The restart set is the last set or the set specified with another **FLDATA32**,REST command. Use a positive value of NSET, ITER, LSTP, or TIME to create a backup of the results file and use a negative value of NSET, ITER, LSTP or TIME if you do not desire a backup of the results file (see Notes below).

Value, *Value2*
Restart point or restart file flag (T or F) as described above.

Fname
File name and directory path of a results file to be used for the restart (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**. This field is valid only with *Label* = NSET, ITER, LSTP, or TIME.

Ext
Filename extension (8 character maximum).

The extension defaults to RFL, and is valid only with *Label* = NSET, ITER, LSTP, or TIME.

--
Unused field

Notes

If both RFIL and WFIL are set to true through the GUI, the state of WFIL will prevail and a new **Jobname.CFD** file will be written.

If the value of NSET, ITER, LSTP, or TIME is positive, the original results file (**Jobname.RFL**) is moved to **Jobname.RFO**, and a new **Jobname.RFL** is created containing all the results sets stored prior to the restart point as well as new results. If the value is negative, the **Jobname.RFL** file will contain the prior results and the new results but the old file will be destroyed. This latter option is used if the results sets currently stored beyond the desired restart point are not worth saving.

If restarting from an existing file other than **Jobname.RFL**, no backup file (**Jobname.RFO**) is created because the existing file is not affected.

If a results file name (*Fname*) is entered for a restart, FLOTRAN interpolates those results onto the current mesh in the database, regardless of whether or not the mesh has changed. This causes the convergence monitors to start again from zero. This restart is different than a restart without a file name specification. However, the results quickly converge to the original solution.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>CFD Restart File

Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Restart/Clear
Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Restart/Iteration
Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Restart/Load step
Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Restart/Set
Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Restart/Time
Main Menu>Preprocessor>FLOTRAN Set Up>Restart Options>Tag set status
Main Menu>Solution>FLOTRAN Set Up>Restart Options>CFD Restart File
Main Menu>Solution>FLOTRAN Set Up>Restart Options>Restart/Clear
Main Menu>Solution>FLOTRAN Set Up>Restart Options>Restart/Iteration
Main Menu>Solution>FLOTRAN Set Up>Restart Options>Restart/Load step
Main Menu>Solution>FLOTRAN Set Up>Restart Options>Restart/Set
Main Menu>Solution>FLOTRAN Set Up>Restart Options>Restart/Time
Main Menu>Solution>FLOTRAN Set Up>Restart Options>Tag set status

FLDATA33, ADVM, *Label*, *Value*

Specifies the approach to discretize the advection term.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

ADVM
Enter the word ADVM in this field.

FLDATA33,ADVM is the **FLDATA** command with its first argument set to ADVM. It can be entered into the program as either **FLDATA33**,ADVM,*Label*,*Value* or **FLDATA**,ADVM,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Specifies the transport equation.

MOME
Momentum equations.

TURB
Turbulence equations.

PRES
Compressible pressure equation.

TEMP
Energy equation.

Value
Choice of approach to discretize the advection term:

MSU
Monotone Streamline Upwind approach (default for PRES).

SUPG
Streamline Upwind / Petrov-Galerkin approach (default for MOME, TURB, and TEMP).

COLG
Collocated Galerkin (COLG) approach.

Notes

See Using SUPG in the *ANSYS Fluids Analysis Guide* for more information on the SUPG approach.

See the *ANSYS, Inc. Theory Reference* for more information on the advection term.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Advection
Main Menu>Solution>FLOTRAN Set Up>Advection

FLDATA34, MIR, *Label*, *Value*

Sets modified inertial relaxation factors.

PREP7: FLOTRAN Stability

MP <> <> <> <> <> <> <> FL PP ED

MIR
Enter the word MIR in this field.

FLDATA34,MIR is the **FLDATA** command with its first argument set to MIR. It can be entered into the program as either **FLDATA34**,MIR,*Label*,*Value* or **FLDATA**,MIR,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Modified relaxation factor labels:

MOME
Momentum modified inertial relaxation.

TURB
Turbulence modified inertial relaxation.

TEMP
Energy modified inertial relaxation.

Value
Modified inertial relaxation factor. *Value* defaults to 0 (modified inertial relaxation off).

Notes

Value must be a positive real number. A *Value* between 0.1 and 1.0 is recommended. A larger *Value* provides a more robust scheme, but it may yield a slower convergence.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>MIR Stabilization
Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>MIR Stabilization

---

FLDATA35, VFTOL, *Label*, *Value*

Specifies tolerances for the lower and upper bound of the volume fraction.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

VFTOL
Enter the word VFTOL in this field.

FLDATA35,VFTOL is the **FLDATA** command with its first argument set to VFTOL. It can be entered into the program as either **FLDATA35**,VFTOL,*Label*,*Value* or **FLDATA**,VFTOL,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Tolerance choices:

VOFL
Lower bound tolerance in the VOF advection algorithm (Value defaults to 1.0e-6).

VOFU
Upper bound tolerance in the VOF advection algorithm (Value defaults to 1.0e-6).

LAML
Lower bound tolerance in the solver for laminar flows (Value defaults to 1.0e-2).

LAMU
Upper bound tolerance in the solver for laminar flows (Value defaults to 1.0e-2).

TRBL
Lower bound tolerance in the solver for turbulent flows (Value defaults to 1.0e-1).

TRBU
Upper bound tolerance in the solver for turbulent flows (Value defaults to 1.0e-1).

Value
Tolerance value for Label above.

Notes

Volume fractions less than the lower bound tolerance are treated as 0. Volume fractions greater than 1 minus the upper bound tolerance are treated as 1.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>VOF Environment>VFRC Tolerance
Main Menu>Solution>FLOTRAN Set Up>VOF Environment>VFRC Tolerance

FLDATA36, AMBV, *Label*, *Value*

Specifies ambient reference values outside of the fluid for the volume of fluid (VOF) method.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

AMBV

Enter the word AMBV in this field.

FLDATA36,AMBV is the **FLDATA** command with its first argument set to AMBV. It can be entered into the program as either **FLDATA36**,AMBV,*Label*,*Value* or **FLDATA**,AMBV,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label

Ambient Reference for which value is being specified:

VX
: U velocity.

VY
: V velocity.

VZ
: W velocity.

TEMP
: Temperature.

ENKE
: Turbulent Kinetic Energy.

ENDS
: Turbulent Dissipation Rate.

Value

Value for the ambient reference as described above.

Notes

PRES is used as a boundary condition at the free surface and for plotting purposes. VX, VY, VZ, TEMP, ENKE and ENDS are only used for plotting purposes.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>VOF Environment>Ambient Condit'n
Main Menu>Solution>FLOTRAN Set Up>VOF Environment>Ambient Condit'n

FLDATA37, ALGR, *Label*, *Value*

Specifies segregated solution or film coefficient algorithms.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

ALGR
Enter the word ALGR in this field.

FLDATA37,ALGR is the **FLDATA** command with its first argument set to ALGR. It can be entered into the program as either **FLDATA37**,ALGR,*Label*,*Value* or **FLDATA**,ALGR,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Algorithm label:

SEGR
Segregated

HFLM
Film Coefficient

Value
If *Label* = SEGR, *Value* specifies the segregated solution algorithm:

SIMPLEF
Original segregated algorithm (default)

SIMPLEN
Enhanced segregated algorithm

If *Label* = HFLM, *Value* specifies the film coefficient algorithm:

MATX
Conductivity matrix algorithm (default)

TEMP
Temperature field algorithm

Notes

Settings automatically changed when SIMPLEN is chosen are not automatically reset if SIMPLEF is reselected. See Coupling Algorithms in the *ANSYS Fluids Analysis Guide* for the settings.

The conductivity matrix algorithm uses the thermal conductivity matrix to calculate heat fluxes and film coefficients. The temperature field algorithm calculates film coefficients directly from thermal gradients.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Algorithm Control
Main Menu>Solution>FLOTRAN Set Up>Algorithm Control

FLDATA38, MASS, *Label*, *Value*

Specifies the mass type for a fluid transient analysis.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

MASS
Enter the word MASS in this field.

FLDATA38,MASS is the **FLDATA** command with its first argument set to MASS. It can be entered into the program as either **FLDATA38**,MASS,*Label*,*Value* or **FLDATA**,MASS,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label
Degree of freedom set for which mass type is being specified:

MOME
Momentum equation.

PRES
Pressure equation.

TURB
Turbulent equation.

TEMP
Energy equation.

ALL
Momentum, pressure, turbulent, and energy equations.

Value
Mass type for fluid transient analysis:

LUMP
Lumped mass matrix (default).

CONS
Consistent mass matrix.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Transient Ctrl>Mass Type
Main Menu>Solution>FLOTRAN Set Up>Transient Ctrl>Mass Type

FLDATA39, REMESH, *Label*, *Value*

Specifies remeshing parameters for transient fluid flow and fluid-solid interaction analyses.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

REMESH
Enter the word REMESH in this field.

FLDATA39,REMESH is the **FLDATA** command with its first argument set to REMESH. It can be entered into the program as either **FLDATA39**,REMESH,*Label*,*Value* or **FLDATA**,REMESH,*Label*,*Value* where *Label* and *Value* are as described below. See the **FLDATA** command for other FLOTRAN CFD input choices.

Label

Remeshing fluid element label (dictates the meaning of *Value*):

ELEM

Value specifies the fluid elements included in remeshing. The following are valid values:

NON

Do not remesh (default).

ALL

Remesh all defined fluid elements if the quality of the worst defined element falls below any quality requirement. Required element qualities are specified by *Label* = ARMAX, VOCH, or ARCH as explained below.

PAR

Remesh defined fluid elements that have a quality below any quality requirement.

A component name may be specified for *Value*. All elements grouped into a component name are remeshed if the quality of the worst element falls below any quality requirement. The component name length is up to 8 characters.

XBNE

Value specifies the elements connected to the boundary nodes excluded from remeshing. The following are valid values:

NONE

Do not exclude any elements connected to the boundary nodes (default).

ALL

Exclude all elements connected to the boundary nodes.

FSI

Exclude elements connected to fluid-solid interaction interfaces.

RESIZ

Value is the element size used for remeshing. *Value* defaults to 0 (the element size at the nearest boundary is used for remeshing).

REXPN

Value is the area expansion (or contraction) option for remeshing. (This option is the same as **SMRTSIZE**,,,EXPND.) This option is used to size internal elements in an area based on the size of the elements on the area's boundaries. *Value* is the expansion (or contraction) factor. For example, issuing **FLDATA39**,REMESH,EXPND,2 before meshing an area will allow a mesh with elements that are approximately twice as large in the interior of an area as they are on the boundary. If *Value* is less than 1, a mesh with smaller elements on the interior of the area will be allowed. *Value* for this option should be greater than 0.5 but less than 4. *Value* defaults to 1, which does not allow expansion or contraction of internal element sizes. If *Value* = 0, the default value of 1 will be used. The actual size of the internal elements will also depend on RESIZ sizing, if used.

ARMA

Value is the maximum allowable element generalized aspect ratio. *Value* defaults to 10.

VOCH
Value is the maximum allowable change of element size (area or volume). *Value* defaults to 3.

ARCH
Value is the maximum allowable element aspect ratio change.*Value* defaults to 3.

STEP
Value is the element quality checking frequency based on time steps. Every *Value* time steps, a quality check takes place. *Value* defaults to 1 (a quality check at every step).

TIME
Value is the only element quality checking time. A quality check takes place at a time specified by *Value*. *Value* defaults to -1 (a quality check at every time point).

Value
Value as described for *Label* above.

Notes

See Table 7.2: "Element Qualities" in the *ANSYS Fluids Analysis Guide* for definitions of element qualities.

Repeat command to set each *Label* as required.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>Thermal Stabilization
Main Menu>Preprocessor>FLOTRAN Set Up>Remesh Ctrl>Element size ctrl
Main Menu>Preprocessor>FLOTRAN Set Up>Remesh Ctrl>Elements for remesh
Main Menu>Preprocessor>FLOTRAN Set Up>Remesh Ctrl>Mesh qualities
Main Menu>Preprocessor>FLOTRAN Set Up>Remesh Ctrl>Remesh frequency
Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>Thermal Stabilization
Main Menu>Solution>FLOTRAN Set Up>Remesh Ctrl>Element size ctrl
Main Menu>Solution>FLOTRAN Set Up>Remesh Ctrl>Elements for remesh
Main Menu>Solution>FLOTRAN Set Up>Remesh Ctrl>Mesh qualities
Main Menu>Solution>FLOTRAN Set Up>Remesh Ctrl>Remesh frequency

---

FLDATA40, WADV, *Label*, *Value*

Controls activation of thermal stabilization near walls.

PREP7: FLOTRAN Turbulence

MP <> <> <> <> <> <> <> FL PP ED

WADV
Enter the word WADV in this field.

FLDATA40,WADV is the **FLDATA** command with its first argument set to WADV. It can be entered into the program as either **FLDATA40**,WADV,*Label*,*Value* or **FLDATA**,WADV,*Label*,*Value* where *Label* and *Value* are as described below.

Label
Enter the word TEMP in this field.

Value
Value controlling *Label*:

TRUE or T
Turn this feature on.

FALSE or F
Turn this feature off.

Notes

Thermal oscillations may occur for turbulent heat transfers in near-wall regions when using the SUPG or the COLG advection scheme with a coarse mesh. Use this command to minimize such spatial oscillations.

For the nodes that lie at the intersection of the wall or solid surface and the inlet, the thermal boundary conditions for those nodes must be the same as the wall or solid surface, not the inlet.

See the **FLDATA33**,ADVM command to see various advection discretization schemes.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Relax/Stab/Cap>Thermal Stabilization
Main Menu>Solution>FLOTRAN Set Up>Relax/Stab/Cap>Thermal Stabilization

---

FLIST, *NODE1*, *NODE2*, *NINC*
Lists force loads on the nodes.

SOLUTION: FE Forces

MP ME ST <> <> PR EM <> <> PP ED

NODE1, *NODE2*, *NINC*
List forces for nodes *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If ALL, list for all selected nodes [**NSEL**] and *NODE2* and *NINC* are ignored (default). If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1*.

Notes

Listing applies to the selected nodes [**NSEL**] and the selected force labels [**DOFSEL**].

Caution: A list containing a node number that is larger than the maximum defined node (*NODE2*), could deplete the system memory and produce unpredictable results.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Forces>On All Nodes
Utility Menu>List>Loads>Forces>On Picked Nodes

FLLIST, *NLOC1*, *NLOC2*, *NINC*

Lists the fatigue location parameters.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

NLOC1, *NLOC2*, *NINC*

List location parameters from *NLOC1* (defaults to 1) to *NLOC2* (defaults to *NLOC1*) in steps of *NINC* (defaults to 1). If *NLOC1* = ALL, *NLOC2* and *NINC* are ignored and all locations are listed.

Menu Paths

Main Menu>General Postproc>Fatigue>List Stress Loc

FLOCHECK, *Key*

Sets up and runs a zero-iteration FLOTRAN analysis.

SOLUTION: FLOTRAN Checkout

MP <> <> <> <> <> <> <> FL PP ED

Key

Determines whether the FLOTRAN analysis is initialized and whether boundary condition status now becomes "old."

0

(Default) Initialization is not performed.

1

Initialization is performed

2

No initialization performed, boundary condition status now becomes "old."

Notes

The **FLOCHECK** command will provide a results summary for the Zeroth iteration (*KEY* = 1) or the current iteration (*KEY* = 0). The results summary provides max/min/average values for each property and DOF. Also, mass flow boundaries are identified and all thermal energy transfer information is summarized. All input information is summarized in the **Jobname.PFL** file (i.e., print file). If you are running the ANSYS program from the GUI, the FLOTRAN print file is echoed to the output window. The **FLOCHECK** command aids verification of boundary condition and property specification.

Initialization [**FLOCHECK**,1] *deletes* any existing **Jobname.PFL** and **Jobname.RFL** files. Run **FLOCHECK** initialization only when you are sure you no longer need the existing results files.

The **FLOCHECK**,2 command changes boundary conditions to the "old" state. It has no effect on the **Jobname.RFL** file. For information on changing fluid boundary conditions, see Applying Transient Boundary Conditions in *ANSYS Fluids Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Flocheck

Main Menu>Solution>FLOTRAN Set Up>Flocheck

FLOTRAN

Specifies "FLOTRAN data settings" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>FLOTRAN Module

FLREAD, *Fname*, *Ext*, --

Reads the residual file written by the FLOTRAN CFD option.

POST1: FLOTRAN Processing

MP <> <> <> <> <> <> <> FL PP ED

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext

Filename extension (8 character maximum).

The extension defaults to RDF (residual file) if *Fname* is blank.

--

Unused field

Menu Paths

Main Menu>General Postproc>Read Results>FLOTRAN 2.1A

FLST, *NFIELD*, *NARG*, *TYPE*, *Otype*, *LENG*

Specifies data required for a picking operation (GUI).

DATABASE: Picking

MP ME ST DY <> PR EM <> FL PP ED

NFIELD

Field number on the command which uses the picking data. (Count the command name as a field, so that a 2 indicates the first command argument, 3 for the second command argument, etc.) The corresponding field on the command will have a P51X label.

NARG

Number of items in the picked list.

TYPE

Type of items picked:

1
Node numbers

2
Element numbers

3
Keypoint numbers

4
Line numbers

5
Area numbers

6
Volume numbers

7
Trace points

8
Coordinate locations (in Global Cartesian coordinates)

9
Screen picks (in X, Y screen coordinates (-1 to 1))

Otype

Data order:

NOOR
Data is not ordered (default).

ORDER
Data is in an ordered list (such as for the **E**,P51X and **A**,P51X commands, in which the order of the data items is significant for the picking operation).

LENG

Length of number of items describing the list (should equal *NARG* if *Otype* = NOOR; default).

Notes

Specifies data required for the **FITEM** command during a picking operation. This is a command generated by the GUI and will appear in the log file (**Jobname.LOG**) if graphical picking is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

On the log file, **FLST** will always be followed by one or more **FITEM** commands which in turn are followed by the ANSYS command that contains a P51X label in one of its fields. This set of commands should not be edited.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

FLUXV

Calculates the flux passing through a closed contour.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

Notes

FLUXV invokes an ANSYS macro which calculates the flux passing through a closed contour (path) predefined by **PATH**. The calculated flux is stored in the parameter FLUX. In a 2-D analysis, at least two nodes must be defined on the path. In 3-D, a path of nodes describing a closed contour must be specified (i.e., the first and last node in the path specification must be the same). A counterclockwise ordering of nodes on the **PPATH** command will give the correct sign on flux. Path operations are used for the calculations, and all path items are cleared upon completion. This macro is only available for vector potential formulations.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>Path Flux

---

FMAGBC, *Cnam1*, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*, *Cnam8*, *Cnam9*

Applies force and torque boundary conditions to an element component.

SOLUTION: Misc Loads

MP ME ST <> <> <> EM <> <> PP ED

Cnam1, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*, *Cnam8*, *Cnam9*

Names of existing element components (**CM** command). Must be enclosed in single quotes (e.g., `Cnam1') when the command is manually typed in.

Notes

FMAGBC invokes a predefined ANSYS macro to apply Maxwell and virtual work force and torque boundary conditions to an element component. These boundary conditions are used for subsequent force and torque

calculations during solution. Magnetic virtual displacements (MVDI = 1) are applied to nodes of elements in the components, and Maxwell surface flags (MXWF) are applied to air elements adjoining the element components. Incorrect force and torque calculations will occur for components sharing adjacent air elements. Companion macros **FMAGSUM** and **TORQSUM** can be used in POST1 to summarize the force and torque calculations, respectively. Torque calculations are valid for 2-D planar analysis only. For 2-D harmonic analysis, force and torque represent time-average values.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>Comp. Force
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>Comp. Force/Torque
Main Menu>Solution>Define Loads>Apply>Electric>Flag>Comp. Force
Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>Comp. Force/Torque

---

FMAGSUM, *Cnam1*, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*, *Cnam8*, *Cnam9*

Summarizes electromagnetic force calculations on element components.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

Cnam1, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*, *Cnam8*, *Cnam9*

Names of existing element components for which Maxwell or virtual work boundary conditions were applied in the preprocessor. Must be enclosed in single quotes (e.g., `Cnam1') when the command is manually typed in.

Notes

FMAGSUM invokes an ANSYS macro that summarizes the Maxwell and virtual work forces. The element components must have had appropriate Maxwell or virtual work boundary conditions established in the preprocessor prior to solution in order to retrieve forces (see **FMAGBC**, **SF**, **BF** commands). The forces are also stored on a per-element basis for the adjacent air layer surrounding the components in the element table [**ETABLE**]. Maxwell forces are stored as items FMX_X, FMX_Y, and FMX_Z, and virtual work forces are stored as items FVW_X, FVW_Y, and FVW_Z. Use the **PLETAB** and **PRETAB** commands to plot and list the element table items.

FMAGSUM can also be used to summarize time-average forces from a 2-D harmonic analysis.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Component Based>Force

---

/FOCUS, *WN*, *XF*, *YF*, *ZF*, *KTRANS*

Specifies the focus point (center of the window).

GRAPHICS: Views

MP ME ST DY <> PR EM <> FL PP ED

WN

Window number (or ALL) to which command applies (defaults to 1).

XF, *YF*, *ZF*
Location of the object to be at the focus point (center of the window) in the global Cartesian coordinate system. If *XF* = AUTO, allow automatic location calculation. If *XF* = USER, use focus location of last display (useful when last display had auto focus).

KTRANS
Translate key:

0
Interpret numerical *XF*, *YF*, *ZF* values as described above.

1
Interpret *XF*, *YF*, *ZF* values as multiples of half-screens to translate from the current position in the screen coordinate system. Example: *XF* of 2.4 translates the display approximately 2.4 half-screens to the left in the screen X (horizontal) direction.

2
Interpret *XF*, *YF*, *ZF* values as multiples of half-screens to translate from the current position in the global Cartesian coordinate system. Example: *XF* of 1.5 translates the display approximately 1.5 half-screens in the global Cartesian X direction of the model.

Command Default

Focus location is automatically calculated to be at the geometric center of the object (modified for centering within the window, depending upon the view).

Notes

Specifies the location on (or off) the model which is to be located at the focus point (center of the window). For section and capped displays, the cutting plane is also assumed to pass through this location (unless the working plane is used via **/CPLANE**). See also **/AUTO** and **/USER** commands.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate
Utility Menu>PlotCtrls>View Settings>Focus Point

FOR2D

Calculates magnetic forces on a body.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

Notes

FOR2D invokes an ANSYS macro which calculates magnetic forces on a body that is completely surrounded by air (symmetry permitted). The calculated forces are stored in the parameters FX and FY. In interactive mode, a node plot is produced with the integration path highlighted. A predefined closed path [**PATH**], passing through the air elements surrounding the body, must be available for this calculation. A counterclockwise ordering of nodes on the **PPATH** command will give the correct sign on the forces. Forces are calculated using a Maxwell

stress tensor approach. The macro is valid for 2-D planar or axisymmetric analysis. Path operations are used for the calculations, and all path items are cleared upon completion.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>Mag Forces

---

FORCE, *Lab*

Selects the element nodal force type for output.

POST1: Controls
POST26: Controls

MP ME ST <> <> PR EM <> <> PP ED

Lab
Type of force to be associated with the force items:

TOTAL
Total forces (static, damping, and inertia).

STATIC
Static forces.

DAMP
Damping forces.

INERT
Inertia forces.

Command Default

Use the total forces.

Notes

FORCE selects the element nodal force type for output with the POST1 **PRESOL**, **PLESOL**, **PRRFOR**, **NFORCE**, **FSUM**, etc. commands, the POST26 **ESOL** command, and reaction force plotting [**/PBC**]. For example, **FORCE**,STATIC causes item F of the **PRESOL** command to be the static forces for the elements processed. Element member forces (such as those available for beams and shells, which are processed by Item and Sequence number) are *not* affected by this command. Damping and inertia forces are only available for full transient and harmonic analyses.

The **PRRSOL** command is not valid with **FORCE**. Use the **PRRFOR** command, which provides the same functionality as **PRRSOL**, instead.

In POST26, the **ESOL** data stored is based on the active **FORCE** specification at the time the data is stored. To store data at various specifications (for example, static and inertia forces), issue a **STORE** command before each new specification.

Menu Paths

Main Menu>General Postproc>Options for Outp

Main Menu>TimeHist Postpro>Define Variables
Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables
Utility Menu>List>Results>Options

FORM, *Lab*

Specifies the format of the file dump.

AUX2: Binary Files

MP ME ST DY <> PR EM <> FL PP ED

Lab

Format:

RECO
: Basic record description only (minimum output) (default).

TEN
: Same as RECO plus the first ten words of each record.

LONG
: Same as RECO plus all words of each record.

Command Default

Basic record description (RECO).

Notes

Specifies the format of the file dump (from the **DUMP** command).

Menu Paths

Utility Menu>File>List>Binary Files
Utility Menu>List>Files>Binary Files

/FORMAT, *NDIGIT*, *Ftype*, *NWIDTH*, *DSIGNF*, *LINE*, *CHAR*

Specifies format controls for tables.

POST1: Listing

MP ME ST DY <> PR EM <> FL PP ED

NDIGIT

Number of digits (3 to 32) in first table column (usually the node or element number). Initially defaults to 7.

Ftype

FORTRAN format types (initially defaults to G):

G
: G*xx*.*yy*. *xx* and *yy* are described below.

F
: F*xx*.*yy*

E
Exx.yy

NWIDTH
Total width (9 to 32) of the field (the *xx* in *Ftype*). Initially defaults to 12.

DSIGNF
Number of digits after the decimal point (*yy* in F or E format) or number of significant digits in G format. Range is 2 to *xx*-7 for *Ftype* = G or E; and 0 to *xx*-4 for *Ftype* = F. Initially defaults to 5.

LINE
Number of lines (11 minimum) per page. Defaults to *ILINE* or *BLINE* from the **/PAGE** command.

CHAR
Number of characters (41 to 240, system-dependent) per line before wraparound. Defaults to *ICHAR* or *BCHAR* from the **/PAGE** command.

Command Default

Program determines format for the data.

Notes

Specifies various format controls for tables printed with the POST1 **PRNSOL**, **PRESOL**, **PRETAB**, **PRRSOL**, and **PRPATH** commands. A blank (or out-of-range) field on the command retains the current setting. Issue **/FORMAT**,STAT to display the current settings. Issue **/FORMAT**,DEFA to reestablish the initial default specifications.

For the POST26 **PRVAR** command, the *Ftype*, *NWIDTH*, and *DSIGNF* fields control the time output format.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

FP, *STITM*, *C1*, *C2*, *C3*, *C4*, *C5*, *C6*
Defines the fatigue S vs. N and Sm vs. T tables.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

STITM
Starting item number for entering properties (defaults to 1). If 1, data input in field *C1* of this command is entered as the first item in the list; if 7, data input in field *C1* of this command is entered as the seventh item in the list; etc. If the item number is negative, *C1*-*C6* are ignored and the item is deleted. If -ALL, the table is erased. Items are as follows (items 41-62 are required only if simplified elastic-plastic code calculations are to be performed):

1,2,...20:
N1, N2, ... N20

21,22,...40:
S1, S2, ... S20

41,42,...50:
T1, T2, ... T10

51,52,...60:
Sm1, Sm2, ... Sm10

61:
M (first elastic-plastic material parameter)

62:
N (second elastic-plastic material parameter)

C1, *C2*, *C3*, *C4*, *C5*, *C6*
Data inserted into six locations starting with *STITM*. If a value is already in one of these locations, it will be redefined. A blank retains the previous value.

Notes

Defines the fatigue alternating stress (S) vs. cycles (N) table and the design stress-intensity value (Sm) vs. temperature (T) table. May also be used to modify any previously stored property tables. Log-log interpolation is used in the S vs. N table and linear interpolation is used in the Sm vs. T table. Cycles and temperatures must be input in ascending order; S and Sm values in descending order. Table values must be supplied in pairs, i.e., every N entry must have a corresponding S entry, etc. Not all property pairs per curve need be used. If no S vs. N table is defined, the fatigue evaluation will not produce usage factor results. See the *ANSYS Structural Analysis Guide* for details.

Menu Paths

Main Menu>General Postproc>Fatigue>Property Table>Elas-plas Par
Main Menu>General Postproc>Fatigue>Property Table>Erase Tables
Main Menu>General Postproc>Fatigue>Property Table>S-N Table
Main Menu>General Postproc>Fatigue>Property Table>Sm_T Table

FPLIST

Lists the property table stored for fatigue evaluation.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

Menu Paths

Main Menu>General Postproc>Fatigue>Property Table>List Tables

---

FREQ, *FREQ1*, *FREQ2*, *FREQ3*, *FREQ4*, *FREQ5*, *FREQ6*, *FREQ7*, *FREQ8*, *FREQ9*
Defines the frequency points for the SV vs. FREQ tables.

SOLUTION: Spectrum Options

MP ME ST <> <> PR <> <> <> PP ED

FREQ1, *FREQ2*, *FREQ3*, *FREQ4*, *FREQ5*, *FREQ6*, *FREQ7*, *FREQ8*, *FREQ9*
Frequency points for SV vs. FREQ tables. Values must be in ascending order. Log-log interpolation will be used between frequency points. *FREQ1* should be greater than zero. Units are cycles/time.

Command Default

No frequency table.

Notes

Repeat **FREQ** command for additional frequency points (20 maximum). Values are added after the last nonzero frequency. If all fields (*FREQ1* -- *FREQ9*) are blank, erase SV vs. FREQ tables.

Frequencies must be in ascending order. Use **STAT** command to list current frequency points. Spectral values are input with the **SV** command and interpreted according to the **SVTYP** command. Applies only to the SPRS (single-point) option of the **SPOPT** command. See the **PSDFRQ** command for frequency input with the MPRS (multi-point) option.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Erase Table
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Freq Table
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Erase Table
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Freq Table

---

FS, *NODE*, *NEV*, *NLOD*, *STITM*, *C1*, *C2*, *C3*, *C4*, *C5*, *C6*
Stores fatigue stress components at a node.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

NODE
Node number corresponding to this location. Used only to associate a node with a new location or to find an existing location.

NEV
Event number to be associated with these stresses (defaults to 1).

NLOD
Loading number to be associated with these stresses (defaults to 1).

STITM
Starting item number for entering stresses (defaults to 1). If 1, data input in field *C1* of this command is entered as the first item in the list; if 7, data input in field *C1* of this command is entered as the seventh item in the list; etc. Items are as follows:

16:
SX, SY, SZ, SXY, SYZ, SXZ total stress components

7:
Temperature

813:
SX, SY, SZ, SXY, SYZ, SXZ membrane-plus-bending stress components.

C1, *C2*, *C3*, *C4*, *C5*, *C6*
Stresses assigned to six locations starting with *STITM*. If a value is already in one of these locations, it will be redefined. A blank retains the previous value (except in the *C1* field, which resets the *STITM* item to zero).

Notes

Stores fatigue stress components at a node as input on this command instead of from the current data in the database. Stresses are stored according to the event number and loading number specified. The location is associated with that previously defined for this node [**FL**] or else it is automatically defined. May also be used to modify any previously stored stress components. Stresses input with this command should be consistent with the global coordinate system for any **FSNODE** or **FSSECT** stresses used at the same location.

Menu Paths

Main Menu>General Postproc>Fatigue>Store Stresses>Specified Val

---

FSAN, *Key*
Turns a fluid-solid interaction analysis on or off.

SOLUTION: Fluid-Structure Interaction

MP <> <> <> <> <> <> <> <> PP ED

Key
Fluid-structure analysis key:

ON
Activates a fluid-solid interaction analysis.

OFF
Deactivates a fluid-solid interaction analysis (default).

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Options

Main Menu>Solution>FSI Set Up>Options

FSCALE, *RFACT*, *IFACT*

Scales force load values in the database.

SOLUTION: FE Forces

MP ME ST <> <> PR EM <> <> PP ED

RFACT
Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

IFACT
Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

Notes

Scales force load (force, heat flow, etc.) values in the database. Scaling applies to the previously defined values for the selected nodes [**NSEL**] and the selected force labels [**DOFSEL**]. Issue **FLIST** command to review results. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

Note — Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

FSCALE does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Forces
Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Forces

FSCO, *Lab*, *VALUE*

Sets convergence values for a fluid-solid interaction analysis.

SOLUTION: Fluid-Structure Interaction

MP <> <> <> <> <> <> <> <> PP ED

Lab
Valid label. Force labels: FX, FY, FZ. Displacement labels: UX, UY, UZ. Temperature label: TEMP. Heat flux label: HFLU. If *Lab* = ALL, **FSCO** applies the convergence value to all eight variables.

VALUE
Convergence value. Defaults to 0.0001.

Notes

FSCO sets convergence values for variables at the fluid-solid interface.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Convergence
Main Menu>Solution>FSI Set Up>Convergence

FSDELE, *NLOC*, *NEV*, *NLOD*

Deletes a stress condition for a fatigue location, event, and loading.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

NLOC
Delete stresses associated with location *NLOC*. Defaults to zero.

NEV
Delete stresses associated with event *NEV*. Defaults to zero.

NLOD
Delete stresses associated with loading NLOD. Defaults to zero.

Notes

Deletes a stress condition stored for a particular fatigue location, event, and loading. Use **FE** command to delete all stresses for a particular event or **FL** command to delete all stresses for a particular location.

Menu Paths

Main Menu>General Postproc>Fatigue>Store Stresses>Dele Stresses

---

FSDT, *INC*

Sets time step increment for a fluid-solid interaction analysis.

SOLUTION: Fluid-Structure Interaction

MP <> <> <> <> <> <> <> <> PP ED

INC
Time step increment. Defaults to 1.

Notes

You cannot use automatic time stepping (**AUTOTS**) in a fluid-solid interaction analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Time
Main Menu>Solution>FSI Set Up>Time

FSIN, *Opt*

Specifies the interface load transfer option for a fluid-solid interaction analysis.

SOLUTION: Fluid-Structure Interaction

MP <> <> <> <> <> <> <> <> PP ED

Opt

Interface load transfer option:

CONS

Conservative formulation for load transfer.

NONC

Nonconservative formulation for load transfer.

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Options
Main Menu>Solution>FSI Set Up>Options

FSIT, *Val1*

Sets the maximum number of stagger iterations for a fluid-solid interaction analysis.

SOLUTION: Fluid-Structure Interaction

MP <> <> <> <> <> <> <> <> PP ED

Val1

Maximum number of iterations. Defaults to 10.

Notes

The maximum number of iterations applies to each time step in the fluid-solid interaction analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Iterations
Main Menu>Solution>FSI Set Up>Iterations

FSLIST, *NLOC1*, *NLOC2*, *NINC*, *NEV*, *NLOD*

Lists the stresses stored for fatigue evaluation.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

NLOC1, *NLOC2*, *NINC*
List stresses from *NLOC1* (defaults to 1) to *NLOC2* (defaults to *NLOC1*) in steps of *NINC* (defaults to 1). If *NLOC1* = ALL, *NLOC2* and *NINC* are ignored and stresses for all locations are listed.

NEV
Event number for stress listing (defaults to ALL).

NLOD
Loading number for stress listing (defaults to ALL).

Notes

Stresses may be listed per location, per event, per loading, or per stress condition. Use **FELIST** and **FLLIST** if only event and location parameters (no stresses) are to be listed.

Menu Paths

Main Menu>General Postproc>Fatigue>Store Stresses>List Stresses

---

FSNODE, *NODE*, *NEV*, *NLOD*

Calculates and stores the stress components at a node for fatigue.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

NODE
Node number for which stress components are stored.

NEV
Event number to be associated with these stresses (defaults to 1).

NLOD
Loading number to be associated with these stresses (defaults to 1).

Notes

Calculates and stores the total stress components at a specified node for fatigue. Stresses are stored according to the event number and loading number specified. The location is associated with that previously defined for this node [**FL**] or else it is automatically defined. Stresses are stored as six total components (SX through SYZ). Temperature is also stored along with the total stress components. Calculations are made from the stresses currently in the database (last ***SET** or **LCASE** command). Stresses stored are in global Cartesian coordinates, regardless of the active results coordinate system [**RSYS**]. The **FSLIST** command may be used to list stresses. The **FS** command can be used to modify stored stresses.

Menu Paths

Main Menu>General Postproc>Fatigue>Store Stresses>From rst File

---

FSOR, *First*, *Second*

Specifies analysis order for a fluid-solid interaction analysis.

SOLUTION: Fluid-Structure Interaction

MP <> <> <> <> <> <> <> <> PP ED

First

First analysis:

FLUID
: Fluid analysis (default).

SOLID
: Solid analysis.

Second

Second analysis:

FLUID
: Fluid analysis.

SOLID
: Solid analysis (default).

Notes

FSOR,SOLID,FLUID can be entered as **FSOR**,SOLID. **FSOR**,FLUID,SOLID can be entered as **FSOR**,FLUID.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Options
Main Menu>Solution>FSI Set Up>Options

FSOU, *Freq*

Sets output frequency for a fluid-solid interaction analysis.

SOLUTION: Fluid-Structure Interaction

MP <> <> <> <> <> <> <> <> PP ED

Freq

Output frequency:

N
: Write solution every Nth (and the last) time step.

ALL
Write solution every time step.

Notes

A **FSOU** setting overrides any other output frequency setting (**OUTRES**, **FLDATA2**, **FLDATA4**, **FLDATA4A**, or **FLDATA5**).

To select the solution items, use the **OUTRES** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Convergence
Main Menu>Solution>FSI Set Up>Convergence

FSPLOT, *NLOC*, *NEV*, *ITEM*

Displays a fatigue stress item for a fatigue location and event.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

NLOC
Display stresses associated with location *NLOC*.

NEV
Display stresses associated with event *NEV*.

ITEM
Display stresses associated with item number *ITEM*. Items are as follows:

16
SX, SY, SZ, SXY, SYZ, SXZ total stress components

7
Temperature

813
SX, SY, SZ, SXY, SYZ, SXZ membrane-plus-bending stress components.

Notes

Displays a fatigue stress item as a function of loading number for a particular fatigue location and event.

Menu Paths

Main Menu>General Postproc>Fatigue>Store Stresses>Plot Stresses

---

FSRE, *Lab*, *VALUE*

Sets relaxation values for a fluid-solid interaction analysis.

SOLUTION: Fluid-Structure Interaction

MP <> <> <> <> <> <> <> <> PP ED

Lab

Valid label. Force label: FORC. Displacement label: DISP. Temperature label: TEMP. Heat flux label: HFLU. Velocity label: VELO. If *Lab* = ALL, **FSRE** applies the relaxation value to all variables.

VALUE

Relaxation value.

Notes

FSRE sets relaxation values for variables at the fluid-solid interface.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Relaxation
Main Menu>Solution>FSI Set Up>Relaxation

FSRS, *Opt*, *VALUE*

Specifies time or load step number for restart of a fluid-structure interaction analysis.

SOLUTION: Fluid-Structure Interaction

MP <> <> <> <> <> <> <> <> PP ED

Opt

Restart option:

TIME

Value is the time for restart.

LDSTEP

Value is the load step number for restart.

VALUE

Time or load step number.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Restart
Main Menu>Solution>FSI Set Up>Restart

FSSECT, *RHO*, *NEV*, *NLOD*, *KBR*

Calculates and stores total linearized stress components.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

RHO

In-plane (X-Y) average radius of curvature of the inside and outside surfaces of an axisymmetric section. If zero (or blank), a plane or 3-D structure is assumed. If nonzero, an axisymmetric structure is assumed. Use a suitably large number (see the *ANSYS, Inc. Theory Reference*) or use -1 for an axisymmetric straight section.

NEV

Event number to be associated with these stresses (defaults to 1).

NLOD

Loading number to be associated with these stresses (defaults to 1).

KBR

For an axisymmetric analysis (RHOp0):

0

Include the thickness-direction bending stresses

1

Ignore the thickness-direction bending stresses

Notes

Calculates and stores the total linearized stress components at the ends of a section path [**PATH**] (as defined by the first two nodes with the **PPATH** command). The path must be entirely within the selected elements (that is, there must not be any element gaps along the path). Stresses are stored according to the fatigue event number and loading number specified. Locations (one for each node) are associated with those previously defined for these nodes [**FL**] or else they are automatically defined. Stresses are separated into six total components (SX through SXZ) and six membrane-plus-bending (SX through SXZ) components. The temperature at each end point is also stored along with the total stress components. Calculations are made from the stresses currently in the database (last ***SET** or **LCASE** command). Stresses are stored as section coordinate components if axisymmetric or as global Cartesian coordinate components otherwise, regardless of the active results coordinate system [**RSYS**]. The **FSLIST** command may be used to list stresses. The **FS** command can be used to modify stored stresses. See also the **PRSECT** and **PLSECT** commands for similar calculations.

Menu Paths

Main Menu>General Postproc>Fatigue>Store Stresses>At Cross Sect

---

FSSOLV, *Fluitit*, *Strutit*, *DIMN*, *Mcomp*, *Xcomp*,*FORCTOL*, *MOMETOL*, *STRUTOL*, *MXLOOP*, *FLUITER*, *RUSEKY*, *RESTKY*

Performs a coupled steady-state fluid-structural analysis.

SOLUTION: Analysis Options

MP <> <> <> <> <> <> <> <> PP ED

Fluitit

Title of the fluid analysis physics file as assigned by the **PHYSICS** command.

Strutit
Title of the structural physics file as assigned by the **PHYSICS** command.

DIMN
Model dimensionality (a default is not allowed):

2
2-D model.

3
3-D model.

Mcomp
Component name of the region to be morphed. For 2-D models, the component may be elements or areas. For 3-D models, the component may be elements or volumes. A component must be specified. You must enclose name-strings in single quotes in the **FSSOLV** command line.

Xcomp
Component name of entities excluded from morphing. In the 2-D case, it is the component name for the lines excluded from morphing. In the 3-D case, it is component name for the areas excluded from morphing. Defaults to exterior non-shared entities (see the **DAMORPH**, **DVMORPH**, and **DEMORPH** commands). You must enclose name-strings in single quotes in the **FSSOLV** command line.

FORCTOL
Fluid force convergence tolerance. Defaults to .005 (0.5%) of the value computed from the previous CFD run. If less than zero, the convergence criteria based on fluid analysis results is turned off.

MOMETOL
Fluid moment convergence tolerance. Defaults to .005 (0.5%) of the value computed from the previous CFD run. If less than zero, the convergence criteria based on fluid analysis results is turned off.

STRUTOL
Structural maximum displacement convergence tolerance. Defaults to .005 (.5%) of the value computed from the previous iteration. If less than zero, the convergence criteria base on structural results is turned off.

MXLOOP
Maximum number of allowable solution recursive loops. A single pass through both a fluid analysis and structural analysis constitutes one loop. Defaults to 100.

FLUITER
Number of global FLOTRAN iterations to be performed on the second and subsequent passes through FLOTRAN during the fluid-structure sequential solution. This option controls the **FLDATA**, ITER, EXEC command. The default value for FLUITER is taken from the value set in the fluid physics file.

RUSEKY
Reuse flag option:

≤ 1
Assumes initial run of **FSSOLV** using base geometry for the first fluid solution.

>1
Assumes **FSSOLV** run is a continuation of a previous **ESSOLV** run, whereby the morphed geometry is used for the initial fluid simulation.

RESTKY
Structural restart key.

0
Use static solution option for structural solution.

1
Use static restart solution option for structural solution.

Notes

FSSOLV invokes an ANSYS macro which automatically performs a coupled fluid-structural analysis. *FORCTOL* and *MOMETOL* represent the tolerance of the total fluid force and moment exerted on the structure. The moment is taken about the global origin. Force and moment are computed using the **INTSRF** command. The macro displays periodic updates of the convergence.

Use *RUSEKY* > 1 for solving multiple **FSSOLV** simulations for different excitation levels. Do not issue the **SAVE** command to save the database between **FSSOLV** calls.

For nonlinear structural solutions, the structural restart option (RESTKY>0) may improve solution time by starting from the previous converged structural solution.

Menu Paths

Main Menu>Preprocessor>Physics>Coupled Solvers>Fluid/struc
Main Menu>Solution>Physics>Coupled Solvers>Fluid/struc

FSSTAT
Lists the settings for a fluid-solid interaction analysis.

SOLUTION: Fluid-Structure Interaction
MP <> <> <> <> <> <> <> <> PP ED

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Status
Main Menu>Solution>FSI Set Up>Status

FSTI, *ENDTIME*, *LOADTIME*
Sets end time and load time for a fluid-solid interaction analysis.

SOLUTION: Fluid-Structure Interaction
MP <> <> <> <> <> <> <> <> PP ED

ENDTIME
Time at end of fluid-solid interaction analysis. Defaults to 1.

LOADTIME
Time to start load transfer for fluid-solid interaction analysis. Defaults to 0.

Notes

A **FSTI** setting overrides any other end time setting (**TIME** or **FLDATA4**).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Time
Main Menu>Solution>FSI Set Up>Time

FSTR, *Field*, *Type*

Specifies static or transient analyses for a fluid-solid interaction analysis.

SOLUTION: Fluid-Structure Interaction

MP <> <> <> <> <> <> <> <> PP ED

Field
Analysis field:

FLUID
Fluid analysis.

SOLID
Solid analysis.

Type
Analysis type:

TRAN
Transient analysis (default).

STAT
Static analysis.

Notes

A **FSTR** specification overrides any other specification of analysis type (**ANTYPE** or **FLDATA1**).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>FSI Set Up>Options
Main Menu>Solution>FSI Set Up>Options

FSUM, *LAB*, *ITEM*

Sums the nodal force and moment contributions of elements.

POST1: Special Purpose

MP ME ST <> <> PR EM <> <> PP ED

LAB

Coordinate system in which to perform summation.

(blank)
Sum all nodal forces in global Cartesian coordinate system (default).

RSYS
Sum all nodal forces in the currently active RSYS coordinate system.

ITEM

Selected set of nodes.

(blank)
Sum all nodal forces for all selected nodes (default), excluding contact elements.

CONT
Sum all nodal forces for contact nodes only.

BOTH
Sum all nodal forces for all selected nodes, including contact elements.

Notes

Sums and prints, in each component direction for the total selected node set, the nodal force and moment contributions of the selected elements attached to the node set. Selecting a subset of nodes [**NSEL**] and then issuing this command will give the total force acting on that set of nodes (default), excluding surface-to-surface contact elements (TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, and CONTA174). Setting *ITEM* = CONT sums the nodal forces and moment contributions of the selected contact elements (surface-to-surface only for CONTA171, CONTA172, CONTA173, and CONTA174). Setting *ITEM* = BOTH sums the nodal forces for all selected nodes, including contact elements. Nodal forces associated with surface loads are not included. The effects of nodal coupling and constraint equations are ignored. Moment summations are about the global origin unless another point is specified with the **SPOINT** command. This vector sum is printed in the global Cartesian system unless it is transformed [**RSYS**] and a point is specified with the **SPOINT** command. By default, the sum is done in global Cartesian, and the resulting vector is transformed to the requested system.

The *LAB* = RSYS option transforms each of the nodal forces into the active coordinate system before summing and printing. The **FORCE** command can be used to specify which component (static, damping, inertia, or total) of the nodal load is to be used. This command output is included in the **NFORCE** command.

This command should not be used with axisymmetric elements.

Using with the NLGEOM Command

If you have activated large deflection (via the **NLGEOM**,ON command), the **FSUM** command generates the following message:

```
Summations based on final geometry and
   will not agree with solution reactions.
```

The message warns that the moment reactions are incorrect. When computing moment reactions, the command assumes that the summation of rotations applies; however, it does *not* apply for large rotations, which require pseudovector representation to sum the rotations.

In contrast, the results for force reactions will be correct because they depend upon linear displacement vectors (which can be added).

Menu Paths

Main Menu>General Postproc>Nodal Calcs>Total Force Sum

FTCALC, *NLOC*, *NODE*

Performs fatigue calculations for a particular node location.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

NLOC
Location number of stress conditions to be used for fatigue calculation.

NODE
Node number (used only for convenience if *NLOC* is not input).

Menu Paths

Main Menu>General Postproc>Fatigue>Calculate Fatig

FTRAN

Transfers solid model forces to the finite element model.

SOLUTION: Solid Forces

MP ME ST <> <> PR EM <> <> PP ED

Notes

Forces are transferred only from selected keypoints to selected nodes. The **FTRAN** operation is also done if the **SBCTRAN** command is issued or automatically done upon initiation of the solution calculations [**SOLVE**].

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Forces
Main Menu>Solution>Define Loads>Operate>Transfer to FE>Forces

---

FTSIZE, *MXLOC*, *MXEV*, *MXLOD*

Defines the fatigue data storage array.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

MXLOC
Maximum number of fatigue locations (defaults to 5).

MXEV
Maximum number of fatigue events (defaults to 10).

MXLOD
Maximum number of loadings in each event (defaults to 3).

Command Default

5 locations, 10 events, 3 loadings.

Notes

Defines the size and erases the stress conditions for the fatigue data storage array. A stress condition is a loading (stresses) at a particular location (node) for a particular event. Size is defined in terms of the maximum number of locations, events, and loadings. The array size cannot be changed once data storage has begun (without erasing all previously stored data). If a size change is necessary, see the **FTWRITE** command.

Menu Paths

Main Menu>General Postproc>Fatigue>Size Settings

FTWRITE, *Fname*, *Ext*, --

Writes all currently stored fatigue data on a file.

POST1: Fatigue

MP ME ST DY <> PR <> <> <> PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to FATG if *Fname* is blank.

--
Unused field

Notes

- Data are written in terms of the equivalent POST1 fatigue commands [**FTSIZE**, **FL**, **FS**, etc.] which you can then edit and resubmit to POST1 (with a **/INPUT** command).
- Once you have created a fatigue data file, each subsequent use of the **FTWRITE** command overwrites the contents of that file.

Menu Paths

Main Menu>General Postproc>Fatigue>Write Fatig Data

/FTYPE, *Ident*, *Type*

Assigns an external or internal type to a binary file.

SESSION: Files

MP ME ST DY <> PR EM <> FL PP ED

Ident

ANSYS file name identifier. Valid identifiers are: EMAT, ESAV, FULL, REDM, SUB, MODE, TRI, DSUB, USUB, EROT, OSAV, and SELD. See the *ANSYS Basic Analysis Guide* for file descriptions. If ALL, apply type to all valid files.

Type

Type key:

EXT
: External type file.

INT
: Internal type file.

Command Default

Type external.

Notes

Binary files may be of type external or internal. Internal files are unformatted and usually use the system's default bit representation of data. External files are unformatted and use 2's complement integer representation and IEEE double precision (64 bit) representation. Some computer systems use the external representation for their internal files. See the *ANSYS Basic Analysis Guide* for details. Binary files with an external representation are transportable between different computer systems, whereas binary files with an internal representation are not.

This command is valid only at the Begin Level.

Menu Paths

Utility Menu>File>ANSYS File Options

FVMESH, *KEEP*

Generates nodes and tetrahedral volume elements from detached exterior area elements (facets).

PREP7: Meshing

MP ME ST DY <> PR EM EH FL PP ED

KEEP

Specifies whether to keep the area elements after the tetrahedral meshing operation is complete.

0

Delete area elements (default).

1

Keep area elements.

Notes

The **FVMESH** command generates a tetrahedral volume mesh from a selected set of detached exterior area elements (facets). (Detached elements have no solid model associativity.) The area elements can be triangular-shaped, quadrilateral-shaped, or a mixture of the two.

The **FVMESH** command is in contrast to the **VMESH** command, which requires a volume to be input.

The main tetrahedra mesher [**MOPT**,VMESH,MAIN] is the only tetrahedra mesher that supports the **FVMESH** command. The alternate tetrahedra mesher [**MOPT**,VMESH,ALTERNATE] does not support **FVMESH**.

Tetrahedral mesh expansion [**MOPT**,TETEXPND,*Value*] is supported for both the **FVMESH** and **VMESH** commands. Tet-mesh expansion is the only mesh control supported by **FVMESH**.

Triangle- or quadrilateral-shaped elements may be used as input to the **FVMESH** command. Where quadrilaterals are used, the default behavior is for the pyramid-shaped elements to be formed at the boundary when the appropriate element type is specified. See the **MOPT**,PYRA command for details.

The **FVMESH** command does not support multiple "volumes." If you have multiple volumes in your model, select the surface elements for one "volume," while making sure that the surface elements for the other volumes are deselected. Then use **FVMESH** to generate a mesh for the first volume. Continue this procedure by selecting one volume at a time and meshing it, until all of the volumes in the model have been meshed.

If an error occurs during the meshing operation, the area elements are kept even if *KEEP* = 0.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Tet Mesh From>Area Elements

G Commands

GAP

Specifies "Reduced transient gap conditions" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Gap Conditions

GAPF, *NVAR*, *NUM*, *Name*

Defines the gap force data to be stored in a variable.

POST26: Set Up

MP ME ST DY <> PR <> <> <> PP ED

NVAR

Arbitrary reference number assigned to this variable (2 to NV [**NUMVAR**]). Overwrites any existing results for this variable.

NUM

Number identifying gap number for which the gap force is to be stored. Issue the **GPLIST** command to display gap numbers.

Name

Thirty-two character name for identifying the item on the printout and displays (defaults to the name **GAPF**).

Notes

Defines the gap force data to be stored in a variable. Applicable only to the expansion pass of the reduced or mode superposition linear transient dynamic (**ANTYPE**,TRANS) analysis. The data usually on ***Fname*.RDSP**.

Menu Paths

Main Menu>TimeHist Postpro>Define Variables
Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

GAPFINISH

Exits from the CAD import topology repair stage.

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

Notes

Data will remain intact in the database, but the database is not automatically written to a file (use the **SAVE** command to write the database to a file).

This command is part of the suite of commands available to find gaps and repair the topology of a model imported from a CAD system through the Default IGES option. These commands are made available if, during a CAD import operation, ANSYS detects topological defects (small gaps, large holes, or overlapping entities). You cannot use the **GAPMERGE** command on a model after issuing the **GAPFINISH** command; however, you can continue to repair the model with the geometry repair commands.

Menu Paths

Main Menu>Preprocessor>Modeling>Topo Repair>Finish

GAPLIST, *Lab*

Lists all joined or disjoined lines in a model (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

Lab
Label defining the type of plot:

OPEN
Lists all disjoined edges. This is the default option.

CLOSED
Lists all seam lines that are currently "sewn" to adjacent areas.

Notes

Disjoined lines are those lines which have only one area attached to them. Typically, these occur around gaps in the model and should be merged with adjacent disjoined lines using the **GAPMERGE** command (topological repair) or **ARFILL** command (geometry repair).

This command is available for both topological and geometric repair for models imported from CAD files through the Default IGES option. If topological repair is active, you can specify what is listed through the **GAPOPT**,OESELE command.

Menu Paths

Main Menu>Preprocessor>Modeling>Geom Repair>Lst Model Gaps>Close Edges
Main Menu>Preprocessor>Modeling>Geom Repair>Lst Model Gaps>Open Edges
Main Menu>Preprocessor>Modeling>Topo Repair>Lst Model Gaps>Close Edges

Main Menu>Preprocessor>Modeling>Topo Repair>Lst Model Gaps>Open Edges

---

GAPMERGE, *Lab*, *VAL1*, *VAL2*, *VAL3*

Merges adjacent disjoined lines (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

Lab

Label defining the type of merge operation.

TOLER

Disjoined lines are merged if they are within the tolerance factor specified by the **GAPOPT** command.

ITER

Disjoined lines are merged through an iterative merging process, where a series of merges occur starting with the tolerance factor specified in *VAL1* until the value specified in *VAL2* is reached. The merges occur at increments specified through *VAL3*. By default, *VAL1* is set to its minimum value (1), *VAL2* is set to its maximum value (10), and *VAL3* is set to 1.

Notes

While the *Lab* = ITER options provides a robust, automatic method for "sewing" gaps, it also requires considerable processing.

This command is part of the suite of commands available to repair the topology of a model imported from CAD systems through the Default IGES option (see Importing Solid Models in the *ANSYS Modeling and Meshing Guide*). These commands are made available if, during a CAD import operation, ANSYS detects topological defects (small gaps, large holes, or overlapping entities). You cannot use the topological repair commands on a model after issuing the **GAPFINISH** command; however, you can continue to repair the model with the geometry commands.

Menu Paths

Main Menu>Preprocessor>Modeling>Topo Repair>Mrg Model Gaps>By Tolerance
Main Menu>Preprocessor>Modeling>Topo Repair>Mrg Model Gaps>Iterative

---

GAPOPT, *Lab*, *Value*

Sets preferences for the CAD import repair commands.

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

Lab

Label Identifying the preference set by the command. The meaning of *Value* varies depending on *Lab*.

TOLER

Set tolerance for the **GAPMERGE** (used only for manual, not iterative, merging), **GAPLIST**, and **GAPPLOT** commands as a factor, defined by *Value*, times the default tolerance. By default, *Value* is set to its minimum value.

OESELE
Determines what will be displayed by the **GAPLIST** and **GAPPLOT** commands. *Value* can be:

ALL
GAPLIST and **GAPPLOT** will display all currently existing disjoined edges with both small gaps and large holes. This is the default option.

MERGE
GAPLIST and **GAPPLOT** will display all open edges that can be merged with the current tolerance.

REMAIN
GAPLIST and **GAPPLOT** will display all open edges that will remain after merging is performed using the current tolerance.

Value
Additional input value described under *Lab*.

Notes

This command is used for the topological repair of models imported from CAD files through the Default IGES option. If no previous defeaturing has been done, setting the tolerance will automatically merge all gaps by the specified value.

Menu Paths

Main Menu>Preprocessor>Modeling>Topo Repair>Preferences

GAPPLOT, *Lab*

Plots all joined or disjoined lines (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

Lab
Label defining the type of plot:

OPEN
Plots disjoined edges. This is the default option.

CLOSED
Plots all edge lines that are currently "sewn" to adjacent areas.

ALL
Plots all line segments (both joined and disjoined). Disjoined lines plot in a different color.

Notes

Disjoined lines are those lines which have only one area attached to them. Typically, these occur around gaps in the model and should be merged with adjacent disjoined lines using the **GAPMERGE** (topological repair) or **ARFILL** (geometry repair) command.

This command is available for both topological and geometric repair for models imported from CAD files through the Default IGES option. If topological repair is active, you can specify what is plotted through the **GAPOPT**,OESELE command.

Menu Paths

Main Menu>Preprocessor>Modeling>Geom Repair>Plt Model Gaps>Closed Edges
Main Menu>Preprocessor>Modeling>Geom Repair>Plt Model Gaps>Open Edges
Main Menu>Preprocessor>Modeling>Geom Repair>Plt Model Gaps>Opn & Closed
Main Menu>Preprocessor>Modeling>Topo Repair>Plt Model Gaps>Closed Edges
Main Menu>Preprocessor>Modeling>Topo Repair>Plt Model Gaps>Open Edges
Main Menu>Preprocessor>Modeling>Topo Repair>Plt Model Gaps>Opn & Closed

---

GAUGE, *Opt*

Gauges the problem domain for an edge-element formulation.

SOLUTION: Analysis Options

MP ME <> <> <> <> EM <> <> PP ED

Opt

Type of gauging performed:

TREE
Perform tree gauging of the edge values (default)

OFF
Do not gauge the edge values

Notes

The **GAUGE** command, required for electromagnetic analyses using an edge potential formulation, gauges the problem domain. The ANSYS program performs gauging over all selected elements and nodes to produce a minimal unique degree of freedom set for solution. By default, gauging is performed at the solver level for each solution [**SOLVE** command or **MAGSOLV** command]. Gauging sets edge values to zero on the tree branches of a graph. The ANSYS program removes these additional constraints after obtaining the solution; therefore, gauging is transparent to users. The **GAUGE**,OFF option is intended for expert ANSYS users who want to create their own gauging instead of using default tree gauging.

For more information about gauging, see the *ANSYS, Inc. Theory Reference*.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>Gauging
Main Menu>Solution>Load Step Opts>Magnetics>Options Only>Gauging

---

/GCMD, *WN*, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*, *Lab10*, *Lab11*, *Lab12*

Controls the type of element or graph display used for the GPLOT command.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which this command applies (defaults to 1)

Lab1, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*, *Lab10*, *Lab11*, *Lab12*
Command labels (for example, **PLNSOL**,S,X)

Notes

This command controls the type of element or graph display that appears when you issue the **GPLOT** command when the **/GTYPE**,,(ELEM or GRPH) entity type is active. If you have multiple plotting windows enabled, you can also use **/GCMD** to select one window when you wish to edit its contents.

For related information, see the descriptions of the **GPLOT** and **/GTYPE** commands in this manual.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Multi-Plot Contrls

/GCOLUMN, *CURVE*, *STRING*

Allows the user to apply a label to a specified curve.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

CURVE
Curve number on which label will be applied (integer value between 1 and 10).

STRING
Name or designation that will be applied to the curve (8 characters max).

Notes

This command is used for an array parameter plot (a plot created by the ***VPLOT** command). Normally the label for curve 1 is "COL 1", the label for curve 2 is "COL 2" and so on; the column number is the field containing the dependent variables for that particular curve. Issuing **/GCOLUMN**,*CURVE*, with no string value specified resets the label to the original value.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Curve

GENOPT

Specifies "General options" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>General Options

GEOM, *K2D*, *NDIV*

Defines the geometry specifications for the radiation matrix calculation.

AUX12: Radiation Substructures

MP ME ST <> <> PR <> <> <> PP ED

K2D

Dimensionality key:

0

3-D geometry

1

2-D geometry (plane or axisymmetric)

NDIV

Number of divisions in an axisymmetric model. Used only with *K2D* = 1. Defaults to 0 (2-D plane). The 2-D model is internally expanded to a 3-D model based on the number of divisions specified ($6 \leq$ *NDIV* ≤ 90). For example, *NDIV* of 6 is internally represented by six 60° sections.

Command Default

3-D geometry.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Other Settings

GEOMETRY

Specifies "Geometry" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Solid Model

/GFILE, *SIZE*

Specifies the pixel resolution on Z-buffered graphics files.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

SIZE

Pixel resolution. Defaults to a pixel resolution of 800. Valid values are from 256 to 2400.

Command Default

800 pixels

Notes

Defines the pixel resolution on subsequently written graphics files (**Jobname.GRPH**) for software Z-buffered displays [**/TYPE**]. Lowering the pixel resolution produces a “fuzzier” image; increasing the resolution produces a “sharper” image but takes a little longer.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Redirect Plots>To GRPH File
Utility Menu>PlotCtrls>Redirect Plots>To PSCR File

/GFORMAT, *Ftype*, *NWIDTH*, *DSIGNF*

Specifies the format for the graphical display of numbers.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

Ftype

FORTRAN format types (G is the default if this field is left blank.)

G

G*xx*.*yy*. *xx* and *yy* are described below.

F

F*xx*.*yy*

E

E*xx*.*yy*

NWIDTH

Total width (12 maximum) of the field (the *xx* in *Ftype*). Defaults to 12.

DSIGNF

Number of digits after the decimal point (*yy* in F or E format) or number of significant digits in G format. Range is 1 to *xx*-6 for *Ftype* = G or E; and 0 to *xx*-3 for *Ftype* = F. The default is a function of *Ftype* and *NWIDTH*.

Notes

Lets you control the format of the graphical display of floating point numbers. Issue **/GFORMAT**,STAT to display the current settings; issue **/GFORMAT**,DEFA to let ANSYS choose the format for the graphical display of floating numbers.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Floating Point Format

/GLINE, *WN*, *STYLE*

Specifies the element outline style.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

WN

Window number (or ALL) to which command applies (defaults to 1).

STYLE

Outline key:

0

Solid element outlines (default)

1
Dashed element outlines

1
No element outlines

Command Default

Solid element outlines

Notes

Determines the element outline style. Often used when node numbers are displayed to prevent element lines from overwriting node numbers.

Unless you are using an OpenGL or Starbase driver, the dashed element outline option (**GLINE**,*WN*,1) is not available in the following situations:

- Z-buffered displays (**/TYPE**,*WN*,6).
- Capped Z-buffered displays (**/TYPE**,*WN*,7).
- Qslice Z-buffered displays (**/TYPE**,*WN*,8).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Edge Options

/GMARKER, *CURVE*, *KEY*, *INCR*

Specifies the curve marking style.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

CURVE
Curve number markers will be applied on (integer value between 1 and 10).

KEY
Marker key:

0
No markers will be applied (default).

1
TRIANGLES will be applied.

2
SQUARES will be applied.

3
DIAMONDS will be applied.

4
CROSSES will be applied.

INCR
Determines the curve marking frequency. (a whole number value between 1 and 255). If *INCR* = 1, markers are displayed at every data point on the curve. If *INCR* = 2 then markers are displayed at every second data point. If *INCR* = 3 then they are displayed at every third data point.

Command Default

No markers will be applied.

Notes

The user-specified markers will not be drawn when the area under the curve is color-filled (**/GROPT**, FILL).

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Curve

GMFACE, *Lab*, *N*

Specifies the facet representation used to form solid models.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

Lab
Valid Labels:

FINE
Value that determines how coarse the facets will be.

N
An integer value between one (small) and ten (large) that determines the tolerances that will be applied to the creation of arcs and surfaces. Ten will create many facets, which may in turn cause ANSYS to run very slowly. One will create fewer facets, which may in turn cause larger tolerance errors.

Menu Paths

Utility Menu>PlotCtrls>Style>Solid Model Facets

/GO

Reactivates suppressed printout.

SESSION: List Controls

MP ME ST DY <> PR EM <> FL PP ED

Notes

Reactivates printout suppressed with the **/NOPR** command without producing any output. The **/GOPR** command has the same function except that it also produces a command response from the program.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Rigid Region
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Section
Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts
Main Menu>Solution>Define Loads>Delete>Structural>Section

/GOLIST

Reactivates the suppressed data input listing.

SESSION: List Controls

MP ME ST DY <> PR EM <> FL PP ED

Notes

Reactivates printout of the data input listing suppressed with **/NOLIST**.

This command is valid in any processor, but only within a batch run [**/BATCH**].

Menu Paths

This command cannot be accessed from a menu.

/GOPR

Reactivates suppressed printout.

SESSION: List Controls

MP ME ST DY <> PR EM <> FL PP ED

Notes

Reactivates printout suppressed with the **/NOPR** command. The **/GO** command has the same function except that it does not produce a command response from the program.

This command is valid in any processor.

Menu Paths

Main Menu>Solution>Time Controls>Time Step Prediction

GP, *NODE1*, *NODE2*, *Lab*, *STIF*, *GAP*, *DAMP*
Defines a gap condition for transient analyses.

SOLUTION: Gap Conditions

MP ME ST <> <> PR <> <> <> PP ED

NODE1
Node I of gap. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NODE2
Node J of gap (must be different from *NODE1*). Non-grounded gap nodes must be defined as master degrees of freedom or be unconstrained, active DOF in a full analysis type. Grounded gap nodes (those not defined as MDOF) need not appear elsewhere in the model.

Lab
Direction of gap action in the nodal coordinate system (implied from the following force labels): FX, FY, FZ, MX, MY, MZ.

STIF
Stiffness (Force/Length) of closed gap (may be positive or negative).

Note — High stiffness requires a small integration time step for numerical stability.

GAP
Initial size of gap. A zero (or positive) value assumes an initially open gap. A negative value defines an interference condition. For a rotational gap, *GAP* should be in radians.

DAMP
Damping coefficient (Force*Time/Length) of closed gap using pseudo velocity (Newmark finite difference expansion scheme).

Notes

Defines a gap condition for the reduced or mode superposition transient (**ANTYPE**,TRANS) analysis. If used in SOLUTION, this command is valid only within the first load step.

Repeat **GP** command for additional gap conditions. Gaps are numbered sequentially as input.

Note — Gaps may be renumbered by the program during the solution (see output listing)

The reduced or mode superposition transient analysis (**ANTYPE**,TRANS and **TRNOPT**,REDUC or MSUP) does not allow gap action with the standard ANSYS gap elements. Gap conditions, however, producing the same effect, may be defined. The gap condition simulates the basic gap action of the COMBIN40 element. The gap condition is treated as an explicit force (equal to the interference times contact stiffness) and affects only the load vector calculation and not the reduced stiffness matrix. The interference is calculated from the displacement extrapolated from the previous time points. A gap condition may be defined between a master degree of freedom and ground or another master degree of freedom. When a non-reduced mode extraction method is used, a master degree of freedom implies an unconstrained, active degree of freedom. Gap nodes not defined as master degrees of

freedom or attached to an element are assumed to be grounded. Grounded gap nodes need not be given a spatial location nor do they need to be located on an element. Gap conditions may be defined in parallel (across the same nodes), with varying gap and stiffness values, to simulate a nonlinear (piecewise) force-deflection curve.

The gap direction is determined from the force label input on the **GP** command, i.e., FX defines a translational gap acting in the UX nodal degree of freedom direction, and MZ defines a rotational gap acting in the nodal ROTZ degree of freedom direction. The actual degree of freedom directions available for a particular node depends upon the degrees of freedom associated with the element types [**ET**] at that node. For example, degrees of freedom available with BEAM3 elements are UX, UY, and ROTZ only (so that only gap labels FX, FY, and MZ are valid).

If the coordinate systems of the nodes connecting the gap are rotated relative to each other, the same degree of freedom may be in different directions. The gap, however, assumes only a one-dimensional action. Nodes I and J may be anywhere in space (preferably coincident). No moment effects are included due to noncoincident nodes. That is, if the nodes are offset from the line of action, moment equilibrium may not be satisfied.

The contact stiffness value represents the stiffness of the closed gap. Stiffness values are related to the integration time step size and should be physically reasonable. High stiffness will require a small integration time step, otherwise, due to the displacement extrapolation, the solution may go unstable. Negative stiffness values may be used with gaps in parallel to produce a decreasing force-deflection curve.

The gap conditions, if any, should be defined in the first load step. Appearances in succeeding load steps are ignored. The order of specifying the gap nodes is important, i.e., a gap condition connecting two nodes will act differently depending upon which node is specified first on the **GP** command. For example, for Node 1 at X = 0.0, Node 2 at X = 0.1, and the gap defined from Node 1 to 2, a displacement of Node 1 greater than Node 2 will cause the gap to close. For the gap defined from Node 2 to 1, a displacement of Node 2 greater than Node 1 will cause the gap to close (like a hook action). In general, the gap closes whenever the separation (defined as U_J - U_I + *GAP*) is negative. U_J is the displacement of node J, U_I is the displacement of node I, and *GAP* is the input gap value. The gap force output appears in the printout only for the time steps for which the gap is closed. A negative spring force is always associated with a closed gap (even with the hook option).

The nonlinear gap damping provided through the DAMP field runs faster than a full transient analysis using a gap element (COMBIN40). Only **ANTYPE** = TRANS and **TRNOPT** = MSUP allow the nonlinear gap damping action. Damping conditions are ignored for the reduced transient analysis method.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Dynamic Gap Cond>Define
Main Menu>Solution>Dynamic Gap Cond>Define

GPDELE, *GAP1*, *GAP2*, *GINC*

Deletes gap conditions.

SOLUTION: Gap Conditions

MP ME ST <> <> PR <> <> <> PP ED

GAP1, *GAP2*, *GINC*

Delete gap conditions from *GAP1* to *GAP2* (defaults to *GAP1*) in steps of *GINC* (defaults to 1).

Notes

Deletes gap conditions defined with the **GP** command. Gap conditions following those deleted are automatically compressed and renumbered. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Dynamic Gap Cond>Delete
Main Menu>Solution>Dynamic Gap Cond>Delete

GPLIST, *GAP1*, *GAP2*, *GINC*

Lists the gap conditions.

SOLUTION: Gap Conditions

MP ME ST <> <> PR <> <> <> PP ED

GAP1, *GAP2*, *GINC*

List gap conditions from *GAP1* to *GAP2* (*GAP2* defaults to *GAP1*) in steps of *GINC* (defaults to 1). If *GAP1* = ALL (default), *GAP2* and *GINC* are ignored and all gap conditions are listed.

Notes

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Dynamic Gap Cond>List All
Main Menu>Preprocessor>Loads>Dynamic Gap Cond>List Specified
Main Menu>Solution>Dynamic Gap Cond>List All
Main Menu>Solution>Dynamic Gap Cond>List Specified
Utility Menu>List>Other>Gap Conditions

GPLOT

Controls general plotting.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Notes

This command displays all entity types as specified via the **/GTYPE** command. Only selected entities (NSEL, ESEL, KSEL, LSEL, ASEL, VSEL) will be displayed. See the descriptions of the **/GTYPE** and **/GCMD** commands for methods of setting the entity types displayed.

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Multi-Plots

/GRAPHICS, *Key*

Defines the type of graphics display.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Key
Graphics key:

FULL
Display all model geometry and results.

POWER
Activate PowerGraphics (default when GUI is on).

Command Default

PowerGraphics ON (*Key* = POWER).

Notes

The **/GRAPHICS** command specifies the type of graphics display. *Key* = POWER activates the PowerGraphics capability. PowerGraphics offers faster plotting than the *Key* = FULL option, and speeds up element, results, area, line, and volume displays.

Results values (both printed and plotted) may differ between the *Key* = FULL and *Key* = POWER options because each option specifies a different set of data for averaging and display. For *Key* = FULL, all element and results values (interior and surface) are included. For *Key* = POWER, only element and results values along the model exterior surface are processed.

Caution: *If you have specified one facet per element edge for PowerGraphics displays (via the **/EFACET** command or via choices from the General Postproc or Utility Menus), PowerGraphics does not plot midside nodes.*

The **/EFACET** command is only applicable to element type displays. (See the descriptions of these commands for more information.)

Maximum values shown in plots can differ from printed maximum values. This is due to different averaging schemes used for plotted and printed maximum values.

PowerGraphics displays do not average at geometric discontinuities. The printouts in PowerGraphics will, however, provide averaging information at geometric discontinuities if the models do not contain shell elements. Carefully inspect the data you obtain at geometric discontinuities.

Note — In Full Graphics mode, it is possible to deselect an individual node, select all elements (including the element that contains that node), and then perform postprocessing calculations on those elements and have that unselected node not be considered in those calculations. However, if PowerGraphics is active, postprocessing always displays based on selected elements.

Commands that are not supported by PowerGraphics are listed below. These commands are executed using the *Key* = FULL option, regardless of whether PowerGraphics is activated. Only certain options for **/CTYPE**, **/EDGE**, **/ESHAPE**, ***GET**, **/PNUM**, **/PSYMB**, **RSYS**, **SHELL**, and ***VGET** are not supported by PowerGraphics. (See the descriptions of these commands for more information.)

/CTYPE	**ESYS**	**/PBF**	**PRETAB**	**SHELL**
DNSOL	***GET**	**PLCRACK**	**PRSECT**	**/SSCALE**
/EDGE	**LAYER**	**PLETAB**	**PRVECT**	**/SHRINK**
ERNORM	**/NORMAL**	**PLLS**	**/PSYMB**	**TALLOW**
ESORT	**NSEL**	**PLSECT**	**RSYS**	***VGET**
	NSORT	**/PNUM**	**SALLOW**	***VPUT**

Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

/GRESUME, *Fname*, *Ext*, --

Sets graphics settings to the settings on a file.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to GSAV if *Fname* is blank.

--
Unused field

Notes

Causes a file to be read to reset the graphics slash (/) commands as they were at the last **/GSAVE** command.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Restore Plot Ctrls

/GRID, *KEY*

Selects the type of grid on graph displays.

GRAPHICS: Graphs

MP ME ST DY <> PR EM <> FL PP ED

KEY

Grid key:

0 (OFF)
No grid.

1 (ON)
Full grid (X and Y grid lines).

2 (X)
Partial grid (X grid lines only).

3 (Y)
Partial grid (Y grid lines only)

Command Default

No grid.

Notes

Selects the type of grid on graph displays. Graphs with multiple Y-axes can have multiple grids [**/GRTYP**]. The grid of the first curve is also used as the background grid (above and behind the curve). Grids for other curves are limited to be under the curves. See also **/GTHK** and **/GROPT** for other grid options.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Grid

/GROPT, *Lab*, *KEY*

Sets various line graph display options.

GRAPHICS: Graphs

MP ME ST DY <> PR EM <> FL PP ED

Lab

Apply display style as selected from the following labels:

AXDV
Axis division (tick) marks (defaults to *KEY* = ON).

AXNM
Axis scale numbers (defaults to *KEY* = ON, which puts numbers at the back plane of the graph). If *KEY* = FRONT, numbers are on the front plane of the graph.

AXNSC
Axis number size scale factor. Input the scale value for *KEY* (defaults to 1.0).

ASCAL
Automatic scaling of additional Y-axes for multi-curve [**/GRTYP**, 2 or 3] graphs (defaults to *KEY* = ON). If *KEY* = OFF, use base Y-axis scaling (see the **/YRANGE** command).

LOGX
Log X scale (defaults to *KEY* = OFF (linear)).

LOGY
Log Y scale (applies only to the base Y axis) (defaults to *KEY* = OFF (linear)).

FILL
Color fill areas under curves (defaults to *KEY* = OFF).

CGRID
Superimpose background grid [**/GRID**] over areas under filled curves (defaults to *KEY* = OFF).

DIG1
Number of significant digits before decimal point for axis values. Input the value for *KEY* (defaults to 4).

DIG2
Number of significant digits after decimal point for axis values. Input the value for *KEY* (defaults to 3).

VIEW
View key for graph displays (defaults to *KEY* = OFF, in which case the view is (0,0,1) for 2-D graph displays or (1,2,3) for 3-D graph displays). If *KEY* = ON, the view settings for graph displays are the same as the view settings for the model.

REVX
Plots the values on the X-axis in reverse order.

REVY
Plots the values on the Y-axis in reverse order.

DIVX
Determines the number of divisions (grid markers) that will be plotted on the X axis.

DIVY
Determines the number of divisions (grid markers) that will be plotted on the Y axis.

LTYP
Determines whether ANSYS generated (*KEY* = 1) or system derived (*KEY* = 0) fonts will be used for the axis labels.

CURL
Determines the position of the curve labels. If (*KEY* = 1), the curve label will be plotted in the legend column, and the label will be displayed in the same color as the curve. If (*KEY* = 0) the curve labels will be plotted near the curve. (default).

XAXO
When you use this label, the subsequent *KEY* value will determine an offset amount from the default (along the bottom) location for the X axis. If *KEY* = 1.0, a full offset occurs (the X axis is moved to the top of the graph). If *KEY* = 0.5, the axis is offset to the midpoint of the graph, and if *KEY* = 0 the axis remains in the original position, along the bottom of the graph. For any offset, a grey copy of the original axis (containing the axis numbering) remains at the original location.

YAXO
When you use this label, the subsequent *KEY* value will determine an offset amount from the default (along the left side of the graph) location for the Y axis. If *KEY* = 1.0, a full offset occurs (the Y axis is moved

to the right side of the graph). If *KEY* = 0.5, the axis is offset to the midpoint of the graph, and if *KEY* = 0 the axis remains in the original position, along the left side of the graph. For any offset, a gray copy of the original axis (containing the axis numbering) remains at the original location.

KEY

Option values:

OFF (0)
Do not apply selected style.

ON (1)
Apply selected style.

nnnn
If *Lab* is DIG1 or DIG2, input the number of digits.

nn
If *Lab* is AXNSC, input the scale factor.

FRONT
If *Lab* is AXNM, FRONT may also be input.

Ndiv
If *Lab* is DIVX or DIVY, determines the number of divisions (1-99) that will be applied to the axis.

Kfont
If *Lab* is LTYP, Kfont is ON (1) or OFF(0). ON will use ANSYS generated fonts for the axis labels, while OFF will use SYSTEM (Windows, X-system, etc.) fonts. The default value is ON (ANSYS fonts).

Notes

Sets various line graph display options. Issue **/GROPT**,STAT to display the current settings. Issue **/GROPT**,DEFA to reset the default specifications. ANSYS informs you that graph view manipulation is inactive unless you have issued the **/GROPT**,VIEW,ON command. See the **/AXLAB**, **/GRTYP**, **/GRID**, and **/GTHK** commands for other graph control options.

Automatic scaling using the **/XRANGE** and **/YRANGE** commands will often yield inappropriate range values for logarithmic scales (**/GROPT**, LOGX or **/GROPT**, LOGY).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes
Utility Menu>PlotCtrls>Style>Graphs>Modify Curve
Utility Menu>PlotCtrls>Style>Graphs>Modify Grid

GRP, *SIGNIF*, *Label*

Specifies the grouping mode combination method.

SOLUTION: Spectrum Options

MP ME ST <> <> PR <> <> <> PP ED

SIGNIF

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single point, multi-point, or DDAM response (**SPOPT**,SPRS, MPRS or DDAM), the significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT**,PSD.)

Label

Label identifying the combined mode solution output.

DISP

Displacement solution (default). Displacements, stresses, forces, etc., are available.

VELO

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

ACEL

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

Notes

The *SIGNIF* value set with this command (including the default value of 0.001) overrides the *SIGNIF* value set with the **MXPAND** command.

This command is also valid for PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine

---

/GRTYP, *KAXIS*

Selects single or multiple Y-axes graph displays.

GRAPHICS: Graphs

MP ME ST DY <> PR EM <> FL PP ED

KAXIS

Axis selection key:

0 or 1
Single Y-axis. Up to 10 curves scaled to a single Y-axis.

2
Additional Y-axes (one for each curve) (3 curves maximum). Allows better scaling of curves with widely differing numbering ranges.

3
Same as 2 but with additional Y-axis and curves projected out of the plane (6 curves maximum). Allows clearer display with an isometric view. The default view when *KAXIS* = 3 is View,1,1,2,3.

Command Default

Single Y-axis graph (except as noted for selection key 3).

Notes

The basic line graph has one or more curves plotted against the same Y and X axes. Multiple curve graphs can also be plotted with individual Y axes and the same X axis. The Y axis of the first curve is referred to as the base Y-axis and the Y axes of the other curves as additional Y axes. Curves are numbered sequentially from 1 (the base curve) in the order in which they are displayed. See the **/AXLAB**, **/GROPT**, **/GRID**, and **/GTHK** commands for other display options.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

/GSAVE, *Fname*, *Ext*, --
Saves graphics settings to a file for later use.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to GSAV if *Fname* is blank.

--
Unused field

Notes

This command does not save all graphics settings, but only those that may be reset by the **/RESET** command. The database remains untouched. Use the **/GRESUME** command to read the file. Repeated use of the **/GSAVE** command overwrites the previous data on the file. The following commands are saved by **/GSAVE**:

/ANGLE	**/DSCALE**	**/GTHCK**	**/PSYMB**	**/VSCALE**
/AXLAB	**/EDGE**	**/GTYPE**	**/RATIO**	**/VUP**
/CLABEL	**/ESHAPE**	**/LIGHT**	**/SHRINK**	**/WINDOW**
/COLOR	**/FOCUS**	**/NORMAL**	**/SSCALE**	**/XRANGE**
/CONTOUR	**/GCMD**	**/NUMBER**	**/TRIAD**	**/YRANGE**
/CPLANE	**/GLINE**	**/PBC**	**/TRLCY**	
/CTYPE	**/GRID**	**/PLOPTS**	**/TYPE**	
/CVAL	**/GROPT**	**/PNUM**	**/VCONE**	
/DIST	**/GRTYP**	**/PSF**	**/VIEW**	

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Save Plot Ctrls

GSBDATA*,LabZ, VALUEZ, LabX, VALUEX, LabY, VALUEY*

Specifies the constraints or applies the load at the ending point for generalized plane strain option.

SOLUTION: FE Constraints

MP ME ST <> <> <> <> <> <> PP ED

LabZ

Constraint or load at the ending point in the fiber Z direction.

F

Apply a force in the fiber direction (default).

LFIBER

Define a length change in the fiber direction.

VALUEZ

Value for *LabZ*. The default is zero.

LabX

Constraint or load on rotation about X.

MX

Supply a moment to cause the rotation of the ending plane about X (default).

ROTX

Define a rotation angle (in radians) of the ending plane about X.

VALUEX

Value for *LabX*. The default is zero.

LabY
Constraint or load on rotation about Y

MY
Supply a moment to cause the rotation of the ending plane about Y (default).

ROTY
Define a rotation angle (in radians) of the ending plane about Y.

VALUEY
Value for *LabY*. The default is zero.

Notes

All inputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see Generalized Plane Strain Option of 18x Solid Elements in the *ANSYS Elements Reference*.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Gen Plane Strain
Main Menu>Solution>Define Loads>Apply>Structural>Gen Plane Strain

---

GSGDATA,*LFIBER*, *XREF*, *YREF*, *ROTX0*, *ROTY0*

Specifies the reference point and defines the geometry in the fiber direction for the generalized plane strain element option.

PREP7: Meshing

MP ME ST <> <> <> <> <> <> PP ED

LFIBER
Fiber length from the reference point. Defaults to 1.

XREF
X coordinate of the reference point. Defaults to zero.

YREF
Y coordinate of the reference point. Defaults to zero.

ROTX0
Rotation of the ending plane about X in radians Defaults to zero.

ROTY0
Rotation of the ending plane about Y in radians Defaults to zero.

Notes

The ending point is automatically determined from the starting (reference) point and the geometry inputs. All inputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see Generalized Plane Strain Option of 18x Solid Elements in the *ANSYS Elements Reference*.

Menu Paths

Main Menu>Preprocessor>Modeling>Genl plane strn

GSLIST,*Lab*

When using generalized plane strain, lists the input data or solutions.

SOLUTION: FE Constraints

MP ME ST <> <> <> <> <> <> PP ED

Lab

Specify the content to be listed.

GEOMETRY
: List the data input using GSGDATA

BC
: List the data input using GSBDATA.

REACTIONS
: When the command is issued in POST1, list the reaction force at the ending point,

  and the moment about X and Y if the corresponding constraints were applied.

RESULTS
: When the command is issued in POST1, list the change of fiber length at the ending point during deformation and the rotation of the ending plane about X and Y during deformation.

ALL
: List all of the above (default).

Notes

This command can be used to list the initial position of the ending plane, the applied load or displacements in the fiber direction, the resulting position of the ending plane after deformation, and the available reaction forces and moments at the ending point.

All inputs and outputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see Generalized Plane Strain Option of 18x Solid Elements in the *ANSYS Elements Reference*.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Other>Genl Plane Strn

GSSOL,*NVAR*, *Item*, *Comp*, *Name*

Specifies which results to store from the results file when using generalized plane strain.

POST26: Set Up

MP ME ST <> <> <> <> <> <> PP ED

NVAR

Arbitrary reference number or name assigned to this variable. Variable numbers can be 2 to *NV* (**NUMVAR**) while the name can be an eight byte character string. Overwrites any existing results for this variable.

Item

Label identifying item to be stored.

LENGTH

Change of fiber length at the ending point.

ROT

Rotation of the ending plane during deformation.

F

Reaction force at the ending point in the fiber direction.

M

Reaction moment applied on the ending plane.

Comp

Component of the item, if Item = ROT or M.

X

The rotation angle or reaction moment of the ending plane about X.

Y

The rotation angle or reaction moment of the ending plane about Y.

Name

Thirty-two character name identifying the item on the printout and display. Defaults to the label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Notes

This command stores the results (new position of the ending plane after deformation) for generalized plane strain. All outputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see Generalized Plane Strain Option of 18x Solid Elements in the *ANSYS Elements Reference*.

Menu Paths

Main Menu>TimeHist Postpro>Variable Viewer

/GST, *Lab*

Turns Graphical Solution Tracking (GST) on or off.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR EM <> FL PP ED

Lab

Determines whether the Graphical Solution Tracking feature is active. Specify ON to activate GST, or OFF to deactivate the feature.

Notes

For interactive runs using the GUI [**/MENU**,ON] or graphics [**/MENU**,GRPH] mode, ANSYS directs GST graphics to the screen. For interactive sessions not using GUI or graphics mode, or for batch sessions, GST graphics are saved in the ANSYS graphics file **Jobname.GST**.

The GST feature is available only for nonlinear structural, thermal, electric, magnetic, fluid, or CFD simulations. For more information about this feature and illustrations of the GST graphics for each analysis type, see the *ANSYS Analysis Guide* for the appropriate discipline. See also the **CNVTOL** command description.

The GST feature is also available for a p-method electrostatic analysis. For interactive sessions with the p-method electrostatic preference set, ANSYS directs the GST graphics to the screen.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Grph Solu Track
Main Menu>Solution>Load Step Opts>Output Ctrls>Grph Solu Track

---

GSUM

Calculates and prints geometry items.

PREP7: Keypoints
PREP7: Lines
PREP7: Areas
PREP7: Volumes

MP ME ST DY <> PR EM <> FL PP ED

Notes

Calculates and prints geometry items (centroid location, moments of inertia, length, area, volume etc.) associated with the selected keypoints, lines, areas, and volumes. Geometry items are reported in the global Cartesian coordinate system. For volumes, a unit density is assumed unless the volumes have a material association via the **VATT** command. For areas, a unit density (and thickness) is assumed unless the areas have a material (and real constant) association via the **AATT** command. For lines and keypoints, a unit density is assumed, irrespective of any material associations [**LATT**, **KATT**, **MAT**]. Items calculated by **GSUM** and later retrieved by a ***GET** or ***VGET** commands are valid only if the model is not modified after the **GSUM** command is issued. This command combines the functions of the **KSUM**, **LSUM**, **ASUM**, and **VSUM** commands.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Geometry

---

/GTHK, *Label*, *THICK*

Sets line thicknesses for graph lines.

GRAPHICS: Graphs

MP ME ST DY <> PR EM <> FL PP ED

Label

Apply thicknesses as selected from the following labels:

AXIS
: Modify thickness of ordinate and abscissa axes on graph displays.

GRID
: Modify thickness of grid lines on graph displays.

CURVE
: Modify thickness of curve lines (when no area fill [**/GROPT**]).

THICK

Thickness ratio (whole numbers only, from -1 to 10):

1
: Do not draw the curve, but show only the markers specified by **/GMARKER**.

0 or 1
: Thin lines.

2
: The default thickness.

3
: 1.5 times the default thickness.

etc.
: (up to 10)

Notes

Sets line thicknesses for graph lines (in raster mode only). Use **/GTHK**,STAT to show settings.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes
Utility Menu>PlotCtrls>Style>Graphs>Modify Curve
Utility Menu>PlotCtrls>Style>Graphs>Modify Grid

/GTYPE, *WN*, *LABEL*, *KEY*

Controls the entities that the GPLOT command displays.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which this command applies (defaults to 1)

LABEL
This represents the type of entity to display:

NODE
Nodes

ELEM
Elements

KEYP
Keypoints

LINE
Lines

AREA
Areas

VOLU
Volumes

GRPH
Graph displays

KEY
Switch:

0
Turns the entity type off.

1
Turns the entity type on.

Notes

The **/GTYPE** command controls which entities the **GPLOT** command displays. NODE, ELEM, KEYP, LINE, AREA, and VOLU are on by default. When ELEM is activated, you can control the type of element displayed via the **/GCMD** command (which also controls the type of graph display). When the GRPH entity type is activated, all other entity types are deactivated. Conversely, when any of the NODE, ELEM, KEYP, LINE, AREA, and VOLU entity types are active, the GRPH entity type is deactivated.

The **/GTYPE** command gives you several options for multi-window layout:

- One window
- Two windows (left and right or top and bottom of the screen)
- Three windows (two at the top and one at the bottom of the screen, or one top and two bottom windows
- Four windows (two at the top and two at the bottom)

Once you choose a window layout, you can choose one of the following: multiple plots, replotting, or no redisplay.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Multi-Plot Contrls

H Commands

HARFRQ, *FREQB*, *FREQE*

Defines the frequency range in the harmonic response analysis.

SOLUTION: Dynamic Options

MP ME ST <> <> PR EM <> <> PP ED

FREQB

Frequency (Hz) at the beginning of the *FREQB* to *FREQE* range (if *FREQE* > *FREQB*). If *FREQE* is blank, the solution is done only at frequency *FREQB*.

FREQE

Frequency at end of this range. Solutions are done at an interval of (*FREQE-FREQB*)*/NSBSTP*, ending at *FREQE*. No solution is done at the beginning of the frequency range. *NSBSTP* is input on the **NSUBST** command. See **EXPSOL** for expansion pass solutions.

Notes

Defines the frequency range for loads in the harmonic response analysis (**ANTYPE**,HARMIC).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Freq and Substps
Main Menu>Solution>Load Step Opts>Time/Frequenc>Freq and Substps

---

/HBC, *WN*, *Key*

Determines how boundary condition symbols are displayed in a display window.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

WN

Window reference number. This number can be any window numbered 1 to 5, or ALL (for all active windows). Defaults to 1

Key

Key to enable/disable hidden surface boundary condition symbol display for 2-D graphics devices and to request improved pressure contour display for 2-D *and* 3-D devices: *Key* = ON, YES or 1 will show your BC symbols on the hidden surfaces and use an improved pressure contour display. *Key* = OFF, NO or 0 (default) will hide the symbols .

Command Default

/HBC,WN,Off

For 2-D graphics devices (X11, win32, PNG, et al.), boundary condition symbols are NOT drawn to the hidden surface display. For both 2-D and 3-D graphics devices, no enhanced rendering of pressure contours occurs.

Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

HBMAT, *fname*, *ext*, --, *form*, *matrx*, *rhs*

Writes an assembled global matrix in Harwell-Boeing format.

AUX2: Binary Files

MP <> <> <> <> <> <> EH <> PP ED

fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Defaults to the current Jobname if left blank.

ext

Output file extension. Defaults to **.matrix** if left blank.

--

Unused fields.

form

Specifies format of output matrix file:

ASCII
: Write output matrix file in ASCII form.

BIN
: Write output matrix file in binary form.

matrx

Specify which matrix to write to the output matrix file:

STIFF
: Write stiffness matrix to output matrix file. Valid for all types of analyses that write a **.FULL** file.

MASS
: Write mass matrix to output matrix file. Valid for buckling, substructure, and modal analyses. If **.FULL** file was generated in a buckling analysis, then this label will write stress stiffening matrix to output matrix file.

DAMP
: Write damping matrix to output matrix file. Only valid for damped modal analyses.

rhs

Specifies whether to write the right-hand side vector to output matrix file

YES
: Write right-hand side vector to output matrix file

NO
: Do not write right-hand side vector to output matrix file

Command Default

By default, assuming a proper filename and extension have been entered, writes the stiffness matrix and right-hand side vector to **jobname.matrix** in the current working directory in ASCII format.

Notes

This command is used to copy a matrix from the assembled global matrix file (**.FULL** file) or from the superelement matrix file (**.SUB** file) as specified on the **FILE** command and write it in Harwell-Boeing format to a new file, named **jobname.MATRIX**. The Harwell-Boeing format is widely used by other applications that deal with matrices.

The assembled global matrix file is created during solution depending on the analysis type, equation solver, and other solution options. By default, the assembled global matrix file is never deleted at the end of solution. For most analysis types, the Sparse direct solver, the ICCG solver, and the AMG solver (when available) will write a **.FULL** file. For buckling and modal analyses, most mode extraction methods will write a **.FULL** file. However, currently only the Block Lanczos and QR-damped mode extraction methods will write a properly formatted **.FULL** file to be used with the **HBMAT** command.

The **WRFULL** command, in conjunction with the **SOLVE** command, can be used to generate the assembled global matrix file and eliminate the equation solution process and results output process.

The Harwell-Boeing format is column-oriented. That is, non-zero matrix values are stored with their corresponding row indices in a sequence of columns. However, since the ANSYS matrix files are stored by row and not column, when the **HBMAT** command is used with a non-symmetric matrix, the transpose of the matrix is, in fact, written.

When dumping the stiffness matrix for transient and harmonic analyses, be aware that the element mass matrix values (and possibly element damping matrix values) are incorporated into the globally assembled stiffness matrix. Thus, the globally assembled stiffness matrix represents more than the stiffness of the model. Please refer to the *ANSYS, Inc. Theory Reference* for more details.

Menu Paths

Utility Menu>File>List>Binary Files
Utility Menu>List>Files>Binary Files

/HEADER, *Header*, *Stitle*, *Idstmp*, *Notes*, *Colhed*, *Minmax*

Sets page and table heading print controls.

POST1: Listing

MP ME ST DY <> PR EM <> FL PP ED

Header

ANSYS page header (system, date, time, version, copyright, title, etc.):

ON
Turns this item on (default for batch mode; not available for interactive mode).

OFF
Turns this item off.

(blank)
Retains the previous setting.

Stitle
Subtitles (see **/STITLE** command): ON, OFF, or (blank) (see above).

Idstmp
Load step information (step number, substep number, time value): ON, OFF, or (blank) (see above).

Notes
Information relative to particular table listings: ON, OFF, or (blank) (see above).

Colhed
Column header labels of table listings (currently only for single column tables): ON, OFF, or (blank) (see above).

Minmax
Minimum/maximum information or totals after table listings: ON, OFF, or (blank) (see above).

Command Default

All specifications are on (batch mode); *Header* = OFF, all other specifications are on (interactive mode)

Notes

Sets specifications on or off for page and table heading print controls associated with the POST1 **PRNSOL**, **PRESOL**, **PRETAB**, **PRRSOL**, and **PRPATH** commands.

Note — If the printout caused a top-of-form (page eject to top of next page), the top-of-form is also suppressed with the printout. Issue **/HEADER**,STAT to display the current settings. Issue **/HEADER**,DEFA to reset the default specifications.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

HELP, *Name*

Displays help information on ANSYS commands and element types.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Name
Command name or element type. Examples are: **HELP**,MP or **HELP**,SOLID45 (or **HELP**,45). For a list of elements of a particular type, enter **HELP**,BEAM, **HELP**,SOLID, **HELP**,HYPER, etc.

Notes

If *Name* uniquely matches a command or element name, the description for that command or element will be displayed in the Help Window. For command help, you must type the complete command name (including the * or /). The help system cannot find partial matches. If * is used at the beginning of the string, it will be interpreted as an ANSYS * command.

For help on topics that are not ANSYS commands or element types (for example, help for the word "material"), use the index or full text search features of the ANSYS online help system.

The **HELP** command is valid only in GUI mode. To obtain help when not in GUI mode, you can either activate the GUI by typing **/MENU**,ON, or you can activate the help system directly by issuing **/UI**,HELP.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

HELPDISP, *Commandname*

Displays help information on DISPLAY program commands.

DISPLAY: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Commandname

Any DISPLAY command. If blank, a list of DISPLAY commands is produced.

Menu Paths

It is part of the DISPLAY command.

HEMIOPT, *HRES*

Specifies options for Hemicube view factor calculation.

SOLUTION: Radiosity

MP ME <> <> <> PR <> <> <> PP ED

HRES

Hemicube resolution. Increase value to increase the accuracy of the view factor calculation. Defaults to 10.

Menu Paths

Main Menu>Preprocessor>Radiation Opts>View Factor
Main Menu>Radiation Opt>Radiosity Meth>View Factor
Main Menu>Solution>Radiation Opts>View Factor

HFANG, *Lab*, *PHI1*, *PHI2*, *THETA1*, *THETA2*

Defines or displays spatial angles of a spherical radiation surface for antenna parameter calculations.

POST1: Special Purpose

MP <> <> <> <> <> <> EH <> PP ED

Lab

Spatial angle label.

ANGLE
Define spatial angles (default).

STATE
Display spatial angles. PHI1, PHI2, THETA1, and THETA2 are ignored.

PHI1, *PHI2*
Starting and ending ϕ angles (degrees) in the spherical coordinate system. Defaults to 0.

THETA1, *THETA2*
Starting and ending θ angles (degrees) in the spherical coordinate system. Defaults to 0.

Notes

Defines or displays spatial angles of a spherical radiation surface. See Spherical Coordinates in the *ANSYS Low-Frequency Electromagnetic Analysis Guide*. Use this command only with **PLHFFAR**,*Opt* = DGAIN or **PRHFFAR**,*Opt* = DGAIN, PRAD, PGAIN, or EFF.

Menu Paths

Main Menu>General Postproc>List Results>Field Extension>Direct Gain
Main Menu>General Postproc>List Results>Field Extension>Efficiency
Main Menu>General Postproc>List Results>Field Extension>Power Gain
Main Menu>General Postproc>List Results>Field Extension>Rad Power
Main Menu>General Postproc>Plot Results>Field Extension>Direct Gain

---

HFARRAY, *NUMX*, *NUMY*, *PX*, *PY*, *SKEW*, *PHASEX*, *PHASEY*

Defines phased array antenna characteristics.

POST1: Special Purpose

MP <> <> <> <> <> <> EH <> PP ED

NUMX
Number of array units in X-direction (defaults to 1).

NUMY
Number of array units in Y-direction (defaults to 1).

PX
Spatial periodicity in X-direction in meters.

PY
Spatial periodicity in Y-direction in meters.

SKEW
Skew angle for triangular periodicity from X-axis to Y-axis in degrees (defaults to 90).

PHASEX
Initial phase angle difference between array units in X-direction in degrees (defaults to 0).

PHASEY
Initial phase angle difference between array units in Y-direction in degrees (defaults to 0).

Menu Paths

Main Menu>General Postproc>List Results>Field Extension>Direct Gain
Main Menu>General Postproc>List Results>Field Extension>Far Field
Main Menu>General Postproc>List Results>Field Extension>Pattern
Main Menu>General Postproc>Plot Results>Field Extension>Direct Gain
Main Menu>General Postproc>Plot Results>Field Extension>Far Field
Main Menu>General Postproc>Plot Results>Field Extension>Pattern

---

HFEIGOPT, *Lab*, *Val1*

Specifies high frequency electromagnetic modal analysis options.

SOLUTION: Analysis Options

MP <> <> <> <> <> <> EH <> PP ED

Lab
High frequency modal analysis type:

CAVITY
Perform a 3-D eigenvalue analysis (default for HF119 and HF120).

CUTOFF
Perform a 2-D cutoff frequency analysis (default for HF118).

GAMMA
Perform a 2-D propagating constant analysis.

Val1
Valid for the GAMMA option only. If Lab = GAMMA, Val1 is the frequency in Hz.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

HFEREFINE, *FACTOR*

Automatically refines high-frequency tetrahedral elements (HF119) or lists high-frequency brick elements (HF120) with the largest error.

PREP7: Special Purpose

MP <> <> <> <> <> <> EH <> PP ED

FACTOR
Factor applied in adaptive mesh refinement scheme. Elements with local error>(FACTOR) (average error) will be refined. FACTOR should be in the range of 1.0 to 3.0 and it defaults to 1.0. Used only with HF119 elements.

NUMLIST
Number of brick elements to be listed. Defaults to 25. Used only with HF120 elements.

Notes

For HF119 elements, **HFEREFINE** automatically refines the mesh so that the measured discretization error will decrease. Execution deletes all boundary conditions and excitation sources on the finite element model.

For HF120 elements, **HFEREFINE** provides a list of elements with the largest error.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>HF Refine

HFNEAR, *Lab*, *VAL*, *X*, *Y*, *Z*, *CS*

Calculates the electromagnetic field at points in the near zone exterior to the equivalent source surface (flagged with the Maxwell surface flag in the preprocessor).

POST1: Special Purpose

MP <> <> <> <> <> <> EH <> PP ED

Lab
Label describing the field:

EF
Electric field (default).

H
Magnetic field.

VAL
VAL is the coordinate system reference number or path.

- *VAL* is the coordinate system reference number. *VAL* may be 0 (Cartesian), 1 (cylindrical), 2 (spherical), 4 or WP (working plane), or any previously defined local coordinate system number (>10). Defaults to 0. **HFNEAR** computes the electromagnetic field for the point X, Y, Z.
- *VAL* = PATH. **HFNEAR** computes the electromagnetic field for the path data points for the path currently defined by the **PATH** and **PPATH** commands.

X, *Y*, *Z*
Location point in the VAL coordinate system. Defaults to 0, 0, 0.

CS
Coordinate system type for calculation of components and magnitude of electric or magnetic field.

0
Cartesian (default).

1
Cylindrical.

2
Spherical.

Notes

HFNEAR uses the equivalent source principle to calculate the electromagnetic field in the near zone exterior to the equivalent source surface (flagged with the Maxwell surface flag in the preprocessor) for either of the following:

- A point X, Y, Z in the *VAL* coordinate system
- A path defined by the **PATH** and **PPATH** commands

To view the electromagnetic field results for a path, use the **PLPAGM** or **PLPATH** commands.

Menu Paths

Main Menu>General Postproc>List Results>Field Extension>Near Field
Main Menu>General Postproc>Path Operations>Map onto Path>HF Near Field

---

HFPA, *Lab*, *Local*, *VAL1*, *VAL2*

Specifies a radiation scan angle for a phased array antenna analysis.

SOLUTION: Analysis Options

MP <> <> <> <> <> <> EH <> PP ED

Lab
Enter SCAN in this field.

Local
Local coordinate system number (defaults to 0).

VAL1
Angle from x-axis towards y-axis, ϕ, in degrees (defaults to 0).

VAL2
Angle from +z-axis towards -z-axis, θ, in degrees (defaults to 0).

Notes

See Figure 4.28: "Spherical Coordinates" in the *ANSYS High-Frequency Electromagnetic Analysis Guide* for an illustration of the coordinate system.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

HFPCSWP, *FREQB*, *FREQE*, *FREQINC*, *Nummode*

Calculates the propagating constants of a transmission line or waveguide over a frequency range.

SOLUTION: Analysis Options

MP ME <> <> <> <> EM <> <> PP ED

FREQB
Beginning frequency of the analysis (in Hertz).

FREQE
Ending frequency of the analysis (in Hertz). If blank, *FREQE* is set to *FREQB* and one solution is performed. (This is recommended for initially verifying the model setup.)

FREQINC
Frequency increment between solutions. If *FREQINC* is blank, solutions are performed at *FREQB* and *FREQE* only (default).

Nummode
Number of required modes. Defaults to 1.

Notes

HFPCSWP calculates the propagating constant for a transmission line or waveguide over a frequency range from *FREQB* to *FREQE*, in increments of *FREQINC*. The output data and graphical displays are written to the files **HFPCSWP.OUT** and **HFPCSWP.GRPH** respectively. Use the DISPLAY program to view the graphs.

HFPCSWP can only be used with HF118 elements.

Menu Paths

Main Menu>Solution>Solve>Electromagnet>HF Emag>2D Freq Sweep

HFPORT, *Portnum*, *Porttype*, *Local*, *Opt1*, *Opt2*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*

Specifies input data for waveguide or transmission line ports.

SOLUTION: Misc Loads

MP <> <> <> <> <> <> EH <> PP ED

Portnum
Waveguide port number. You specify the integer number for exterior and interior waveguide ports by the **SF** and **BF** family of commands, respectively. The number must be between 1 and 50. If Status, provide status of port option settings.

Porttype
Waveguide type:

COAX
Cylindrical coaxial waveguide.

RECT
Rectangular waveguide.

CIRC
Circular waveguide.

PARA
Parallel plate waveguide.

TLINE
Transmission line.

Local
A previously defined local Cartesian coordinate system number (>10) or 0 (global Cartesian) used to specify the geometric properties of the waveguide. Defaults to 0. The local Z-direction must be the direction of propagation.

Opt1
For *Porttype* = COAX, RECT, CIRC, PARA, *Opt1* = Mode type:

TEM
Transverse electromagnetic wave.

TEmn
Transverse electric wave. See notes below.

TMmn
Transverse magnetic wave. See notes below.

For *Porttype* = TLINE, *Opt1* = Path name for calculating the voltage drop across the port (no default). The path name is limited to a maximum of 8 characters. (For information about defining paths, see the **PATH** command description).

Opt2
For *Porttype* = COAX, RECT, CIRC, PARA, *Opt2* = Port boundary condition:

IMPD
Wave impedance matching condition of single propagating mode. Valid for exterior port.

SEXT
S-parameter extraction port. Valid for interior ports.

SOFT
Port used for launching a soft source. Valid for interior ports.

HARD
Port used for launching a hard source. Valid for exterior or interior ports.

For *Porttype* = TLINE, *Opt2* = SEXT for a S-parameter extraction port.

VAL1, *VAL2*, *VAL3*, *VAL4*, *VAL5*
If *Porttype* = COAX:

VAL1
Inner radius of the coaxial waveguide.

VAL2
Outer radius of the coaxial waveguide.

VAL3
Zero-to-peak amplitude of the voltage between the inner and the outer conductors. If blank, the port will create a matching impedance.

VAL4
Phase angle of the applied voltage (in degrees). Defaults to 0 degrees.

VAL5
Input power (time-average). If power is input, it overrides the applied voltage input.

If *Porttype* = RECT:

VAL1
Width of the rectangular waveguide

VAL2
Height of the rectangular waveguide.

VAL3
Zero-to-peak amplitude of the electric field component E_z for a TM wave or the magnetic field component H_z for a TE wave. If blank, the port will appear as a matching impedance.

VAL4
Phase angle of the applied field (in degrees). Defaults to 0 degrees.

VAL5
Input power (time-average). If power is input, it overrides the field component input.

If *Porttype* = CIRC

VAL1
Radius of the circular waveguide.

VAL2
Not used.

VAL3
Zero-to-peak amplitude of the electric field component E_z for a TM wave or the magnetic field component H_z for a TE wave. If blank, the port will appear as a matching impedance.

VAL4
Phase angle of the applied field (in degrees). Defaults to 0 degrees.

VAL5
Input power (time-average). If power is input, it overrides the field component input.

If *Porttype* = PARA:

VAL1
Width of the parallel plate waveguide (defaults to 1).

VAL2
Separation between the two plates.

VAL3
Zero-to-peak amplitude of the electric field component E_y for a TEM wave, electric field component E_z for a TM wave, or the magnetic field component H_z for a TE wave. If blank, the port will appear as a matching impedance.

VAL4
Phase angle of the applied field (in degrees). Defaults to 0 degrees.

VAL5
Input power (time-average). If power is input, it overrides the field component input.

If *Porttype* = TLINE:

VAL1
Resistance of reference impedance (in Ohms) (defaults to 50).

VAL2
Susceptance of reference impedance (in Ohms) (defaults to 0).

VAL3
Not used.

VAL4
Not used.

VAL5
Not used.

Notes

The origin of the local coordinate system must be at the center of the waveguide structure. For a rectangular waveguide, the X and Y axes of the local coordinate system must be parallel to the width and height of the waveguide, respectively. For a parallel plate waveguide, the X and Y axes of the local coordinate system must be parallel to the width and separation of the waveguide, respectively.

The following apply to the TEmn and TMmn mode types:

- For a rectangular waveguide, the suffix m and n mean the variation of the field along the wide side and narrow side of the waveguide, respectively.
- For a circular waveguide, the suffix m and n mean the variation of the field along the angular and radial directions, respectively.
- For a parallel plate waveguide, the suffix m is 0 and the suffix n means the variation of the field between the plates.
- For a coaxial waveguide, only the TEM mode is available.

For transmission line ports (*Porttype* = TLINE), the voltage path should be defined on the port plane. The excitation plane should be defined a short distance away from the transmission line port if the excitation source is defined by EF, JS, or H excitation via the **BF** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Modify Port
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Port Status
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Interior Port>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Modify Port
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Port Status

---

HFSCAT, *Lab*

Specifies a high-frequency scattering analysis.

SOLUTION: Analysis Options

MP <> <> <> <> <> <> EH <> PP ED

Lab
Label identifying scattering analysis options:

OFF
Do not perform a scattering analysis.

SCAT
Perform a scattering analysis and store the scattering field (default).

TOTAL
Perform a scattering analysis and store the total field.

Notes

Specifies a high frequency scattering analysis and the type of electromagnetic field output. **HFSCAT**,SCAT provides a scattering field output, E^{sc}, which is required for the calculation of Radar Cross Section (RCS). **HFSCAT**,TOTAL provides a sum of the incident and scattering fields, $E^{total} = E^{inc}+E^{sc}$.

Use the **PLWAVE** command to specify the incident field, E^{inc}.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

HFSWEEP, *FREQB*, *FREQE*, *FREQINC*, *Portin*, *Port2*, *Port3*, *Port4*, *Pvolt*, *Pang*, *Pdist*, *Vpath*, *Ipath*, *Vsymm*, *Isymm*

Performs a harmonic response for a high-frequency electromagnetic wave guide analysis.

SOLUTION: Analysis Options

MP ME <> <> <> <> EM <> <> PP ED

FREQB
Beginning frequency of the analysis (in Hertz).

FREQE
Ending frequency of the analysis (in Hertz). If blank, *FREQE* is set to *FREQB* and one solution is performed. (This is recommended for initially verifying the model setup.)

FREQINC
Frequency increment between solutions. The analysis performs solutions at *FREQB*, *FREQE*, and equidistant solutions between these frequencies. If *FREQINC* is blank, the ANSYS program performs solutions at *FREQB* and *FREQE* only (default).

Portin
Port number of the input (excited) port with a TE10 mode or COAX mode excitation. (See the **HFPORT** and **SPARM** command descriptions.) If *Portin* is blank (default), ANSYS does no S-parameter or reflection coefficient calculations.

Port2
Output port number of a matched port. The S parameters for *Portin* and *Port2* are calculated. If *Port1* is blank (default), ANSYS does no S-parameter calculations and ignores input values for *Port2* and *Port3*.

Port3
Second output port number of a matched port.

Port4
Third output port number of a matched port.

Pvolt
Port EMF (voltage drop magnitude) applied to the excited port. This is required for reflection coefficient calculations.

Pang
Phase angle (in degrees) of the port EMF (voltage drop). Defaults to zero.

Pdist
Propagation distance between the excited port and the evaluation point. If the *Pdist* value is zero, the reflection coefficient, VSWR, and return loss at the location will be calculated at the excited port. (See the **REFLCOEF** command description.)

Vpath
Path name for a predefined path for calculating the EMF (voltage drop) between two points. (For information about defining paths, see the **PATH** command description.) Use the **PASAVE** command to save paths after defining them. For calculating reflection coefficients in a COAX conductor, the path must be located at the propagation distance from the input port and must extend from the inner conductor to the outer conductor. (See the **REFLCOEF** command description for more information.) For an impedance calculation, the path may be located anywhere. If specified along with an MMF (current) path (see *Ipath* below), impedance will be calculated. (See the **IMPD** command description.)

Ipath
Path name for a predefined path, enclosing a conductor, for calculating MMF (current). The path should traverse a closed contour surrounding the conductor, and should be defined in a counterclockwise directions. If you specify *Ipath* with *Vpath*, ANSYS calculates the port impedance. Use the **PASAVE** command to save paths after defining them.

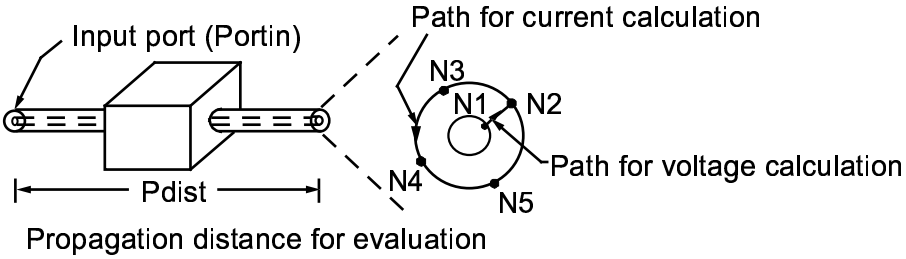
Vsymm
Symmetry factor applied to the calculated voltage drop. *Vsymm* specifies the voltage drop from the conductor to the reference point and is used in the port impedance calculation.

Isymm
Symmetry factor applied to the calculated current. The calculated current is multiplied by *Isymm*, and is used in the port impedance calculation.

Notes

HFSWEEP performs a harmonic response for a high-frequency electromagnetic waveguide analysis over a frequency range from *FREQB* to *FREQE*, in increments of *FREQINC*. **HFSWEEP** calculates S-parameters at each frequency step for each port. For a COAX mode waveguide (see the **HFPORT** command description), reflection coefficients may be calculated in a similar manner at a specified distance (*Pdist*) from the input port. You also can calculate impedance at a specified distance. The ANSYS program writes output data and graphical displays to files **HFSWEEP.OUT** and **HFSWEEP.GRPH** respectively. Use the DISPLAY program to view the graphs.

The diagram below shows an example waveguide and the paths used for MMF (current) and EMF (voltage drop) calculations:



Menu Paths

Main Menu>Solution>Solve>Electromagnet>HF Emag>Freq Sweep

HFSYM, *KCN*, *Xkey*, *Ykey*, *Zkey*

Indicates the presence of symmetry planes for the computation of high-frequency electromagnetic fields in the near and far field domains (beyond the finite element region).

POST1: Special Purpose

MP <> <> <> <> <> <> EH <> PP ED

KCN

Coordinate system reference number. *KCN* may be 0 (Cartesian), or any previously defined local Cartesian coordinate system number (>10). Defaults to 0.

Xkey

Key for electromagnetic field boundary condition, as prescribed for the solution, corresponding to the x = constant plane:

None
: No electric wall or magnetic wall boundary conditions (default).

PEC
: Perfect electric conductor (electric wall boundary conditions).

PMC
: Perfect magnetic conductor (magnetic wall boundary conditions).

Ykey

Key for electromagnetic field boundary condition, as prescribed for the solution, corresponding to the y = constant plane:

None
No electric wall or magnetic wall boundary conditions (default).

PEC
Perfect electric conductor (electric wall boundary conditions).

PMC
Perfect magnetic conductor (magnetic wall boundary conditions).

Zkey
Key for electromagnetic field boundary condition, as prescribed for the solution, corresponding to the z = constant plane:

None
No electric wall or magnetic wall boundary conditions (default).

PEC
Perfect electric conductor (electric wall boundary conditions).

PMC
Perfect magnetic conductor (magnetic wall boundary conditions).

Notes

HFSYM uses the image principle to indicate symmetry planes (x, y, or z = constant plane) for high-frequency electromagnetic field computations outside the modeled domain. A perfect magnetic conductor (PMC) boundary condition must be indicated even though it occurs as a natural boundary condition.

Menu Paths

Main Menu>General Postproc>List Results>Field Extension>Direct Gain
Main Menu>General Postproc>List Results>Field Extension>Efficiency
Main Menu>General Postproc>List Results>Field Extension>Far Field
Main Menu>General Postproc>List Results>Field Extension>Near Field
Main Menu>General Postproc>List Results>Field Extension>Pattern
Main Menu>General Postproc>List Results>Field Extension>Power Gain
Main Menu>General Postproc>List Results>Field Extension>Rad Power
Main Menu>General Postproc>List Results>Field Extension>RCS
Main Menu>General Postproc>List Results>Field Extension>RCS Normalized
Main Menu>General Postproc>Path Operations>Map onto Path>HF Near Field
Main Menu>General Postproc>Plot Results>Field Extension>Direct Gain
Main Menu>General Postproc>Plot Results>Field Extension>Far Field
Main Menu>General Postproc>Plot Results>Field Extension>Pattern
Main Menu>General Postproc>Plot Results>Field Extension>RCS
Main Menu>General Postproc>Plot Results>Field Extension>RCS Normalized

HMAGSOLV, *FREQ*, *NRAMP*, *CNVA*, *CNVV*, *CNVC*, *CNVE*, *NEQIT*

Specifies 2-D or axisymmetric harmonic magnetic solution options and initiates the solution.

SOLUTION: Analysis Options

MP ME ST <> <> <> EM <> <> PP ED

FREQ
Analysis frequency (Hz).

NRAMP
Number of ramped substeps for the first load step of a nonlinear 2-D harmonic electromagnetic solution. Defaults to 3. If *NRAMP* = -1, ignore the ramped load step entirely.

CNVA
Convergence tolerance on the program calculated reference value for the magnetic vector potential degree of freedom. Defaults to 0.001.

CNVV
Convergence tolerance on the program calculated reference value for the time-integrated electric potential VOLT. Defaults to 0.001.

CNVC
Convergence tolerance on the program calculated reference value for the current degree of freedom CURR. Defaults to 0.001.

CNVE
Convergence tolerance on the program calculated reference value for the voltage drop degree of freedom EMF. Defaults to 0.001.

NEQIT
Maximum number of equilibrium iterations per load step. Defaults to 50.

Notes

HMAGSOLV invokes an ANSYS macro which specifies harmonic electromagnetic solution options and initiates the solution. The macro is applicable to any ANSYS 2-D or axisymmetric linear or nonlinear harmonic analysis. Results are only stored for the final converged solution. (In POST1, issue ***SET**,LIST to identify the load step of solution results.) The macro internally determines if a nonlinear analysis is required based on magnetic material properties defined in the database.

The macro performs a two-load-step solution sequence. The first load step ramps the applied loads over a prescribed number of substeps (*NRAMP*), and the second load step calculates the converged solution. For linear problems, only a single load step solution is performed. The ramped load step can be bypassed by setting *NRAMP* to -1.

A 3-D harmonic electromagnetic analysis is available for linear solutions only and does not require this solution macro.

The following analysis options and nonlinear options are controlled by this macro: **KBC**, **NEQIT**, **NSUBST**, **CNVTOL**, **OUTRES**.

Menu Paths

Main Menu>Solution>Solve>Electromagnet>Harmonic Analys>Opt&Solv

HPGL, *Kywrd*, *Opt1*, *Opt2*

Specifies various HP options.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

If *Kywrd* = **MODEL**, command format is **HPGL**,MODEL,*Pmod*.

Pmod
Valid plotter model: 7475A (default), 7550A, 7580B, 7585B, 7586B, COLORPRO, DRAFTPRO, or DRAFTMASTER.

If *Kywrd* = **PAPER**, command format is **HPGL**,PAPER,*Size*,*Orien*.

Size
Valid paper size: A (default), B, C, D, E, A4, A3, A2, A1, A0, CARCH, DARCH, or EARCH.

Orien
Orientation: HORIZONTAL (default) or VERTICAL.

If *Kywrd* = **COLOR**, command format is **HPGL**,COLOR,*KEY*.

KEY
Pen choice:

0
Single pen

1
Multiple pens

If *Kywrd* = **DIRECT**, command format is **HPGL**,DIRECT,*Port*.

Used to direct plotter commands to a port. Use **HPGL**,FILE to redirect output from a port back to a file. This option for *Kywrd* is valid in the DISPLAY program only.

Port
Port name for direct connection.

Notes

This command is available in both the ANSYS and DISPLAY programs. It is valid for Hewlett Packard Graphics Language (HPGL) format files selected in the ANSYS program with **/SHOW**,HPGL (or HPGL2), or with **/SHOWDISP**,HPGL (or HPGL2) in the DISPLAY program.

An output file is generated for each plot. The ANSYS file is named **Jobname*NN*.HPGL**. In the DISPLAY program, this file is named **HPGL*nn***. This file remains open for a subsequent **/NOERASE** plot, and will be incomplete until the program is closed (**/EXIT**), or until the next file is opened by the next **/ERASE** plot request.

Menu Paths

Utility Menu>PlotCtrls>Redirect Plots>To HPGL File
Utility Menu>PlotCtrls>Redirect Plots>To HPGL2 File

HPTCREATE, *TYPE*, *ENTITY*, *NHP*, *LABEL*, *VAL1*, *VAL2*, *VAL3*

Defines a hard point.

PREP7: Hard Points

MP ME ST DY <> PR EM <> FL PP ED

TYPE
Type of entity on which the hard point will be created.

LINE
Hard point will be created on a line.

AREA
Hard point will be created within an area (not on the boundaries).

ENTITY
Number of the line or area on which the hard point will be created.

NHP
Number assigned to the hard point. Defaults to the lowest available hard point number.

LABEL
If *LABEL* = COORD, *VAL1*, *VAL2*, and *VAL3* are the respective global X, Y, and Z coordinates. If *LABEL* = RATIO, *VAL*1 is the parameter value (this is available only for lines). Valid parameter values are between 0 and 1. *VAL2* and *VAL3* are ignored.

VAL1
If *LABEL* = RATIO, ratio value for line. If *LABEL* = COORD, global X coordinate value.

VAL2
If *LABEL* = COORD, global Y coordinate value.

VAL3
If *LABEL* = COORD, global Z coordinate value.

Notes

The ability to enter a parameter value provides a simple way of positioning hard points on lines. For example, to place a hard point halfway along a line, one can simply specify a *VAL1* value of 0.5.

For models imported through the DEFAULT IGES filter, you can place hard points on models only by specifying coordinates (you can't place a hard point using interactive picking).

If you issue any commands that update the geometry of an entity, such as Boolean or simplification commands, any hard points associated with that entity are deleted. Therefore, you should add any hard points after completing the solid model. If you delete an entity that has associated hard points, those hard points are either

- Deleted along with the entity (if the hard point is not associated with any other entities).
- Detached from the deleted entity (if the hard point is associated with additional entities).

When archiving your model (**CDWRITE**), hardpoint information cannot be written to the IGES file. The **Jobname.cdb** file can be written with the **CDWRITE**,DB option.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Hard Points>Hard PT on area>Hard PT by coordinates
Main Menu>Preprocessor>Modeling>Create>Hard Points>Hard PT on line>Hard PT by coordinates
Main Menu>Preprocessor>Modeling>Create>Hard Points>Hard PT on line>Hard PT by ratio
Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on area>Hard PT by coordinates
Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on area>Hard PT by picking
Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on line>Hard PT by coordinates
Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on line>Hard PT by picking
Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on line>Hard PT by ratio

---

HPTDELETE, *NP1*, *NP2*, *NINC*

Deletes selected hardpoints.

PREP7: Hard Points

MP ME ST DY <> PR EM <> FL PP ED

NP1, *NP2*, *NINC*
Delete the pattern of hard points beginning with *NP1* to *NP2* in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and the pattern is all selected hard points [**KSEL**]. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored.

Notes

Deletes all attributes attached to the designated hard points as well as the hard points themselves. If any entity is attached to a designated hard point, the command detaches the hard point from that entity (the program will alert you that this will occur).

Menu Paths

Main Menu>Preprocessor>Modeling>Delete>Hard Points

HRCPLX, *LOADSTEP*, *SUBSTEP*, *OMEGAT*, *1STLCASE*, *2NDLCASE*

Computes and stores in the database the time-harmonic solution at a prescribed phase angle.

POST1: Set Up

MP ME ST DY <> PR EM <> FL PP ED

LOADSTEP
Load step number of the data set to be read (defaults to 1).

SUBSTEP
Substep number within *LOADSTEP*.

OMEGAT
Phase angle defined by the product of the angular frequency and time. If set to 0.0, real results are supplied. If set to -90.0, imaginary results are supplied. If set to 90.0, imaginary results are supplied. If set >= 360, amplitude is supplied. All others supply results at that phase angle.

1STLCASE
First load case number (defaults to 1).

2NDLCASE
Second load case number (defaults to 2).

Notes

HRCPLX combines the real and imaginary parts of results data to give the "total" solution at the specified phase angle. *1STLCASE* and *2NDLCASE* are internally generated load cases. You may want to specify these to avoid overwriting an existing load case number 1 or 2. Not all results computed from this command are valid. See Summable, Non-summable, and Constant Data in the *ANSYS Basic Analysis Guide* for more information.

Since **HRCPLX** performs load case combinations, it alters most of the data in the database. In particular, it alters applied loads such as forces and imposed displacements. To restore the original loads in the database for a subsequent analysis, reissue the **SET** command in POST1 to retrieve the real and imaginary set data.

See the *ANSYS, Inc. Theory Reference* for more information on harmonic analysis equations and their relationship to real and imaginary data sets.

Menu Paths

This command cannot be accessed from a menu.

HREXP, *ANGLE*

Specifies the phase angle for the harmonic analysis expansion pass.

SOLUTION: Dynamic Options

MP ME ST <> <> PR <> <> <> PP ED

ANGLE
Phase angle (degrees) for expansion pass. If ALL (default), use both 0.0° (real) and 90.0° (imaginary) phase angles.

Notes

Specifies the phase angle where the expansion pass will be done for a harmonic reduced or harmonic mode superposition expansion pass.

This command is ignored if the **HROPT** command has been issued with *Method* = SX or *Method* = SXRU.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Load Step
Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq
Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's
Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Load Step
Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq
Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's

---

HROPT, *Method*, *MAXMODE*, *MINMODE*, *MCout*, *Damp*

Specifies harmonic analysis options.

SOLUTION: Dynamic Options

MP ME ST <> VT PR EM EH <> PP ED

Method

Solution method for the harmonic analysis:

FULL
Full method (default).

REDUC
Reduced method.

MSUP
Mode superposition method.

SX
Variational Technology

SXRU
Variational Technology reusing the analysis results generated in a previous analysis using *Method* = SX.

MAXMODE

Largest mode number to be used to calculate the response (for *Method* = MSUP). Defaults to the highest mode calculated in the preceding modal analysis. This option does not apply for *Method* = SX and *Method* = SXRU.

MINMODE

Smallest mode number to be used (for *Method* = MSUP). This option does not apply for *Method* = SX and *Method* = SXRU. Defaults to 1.

MCout

Modal coordinates output key (valid only for the mode superposition method MSUP):

NO
No output of modal coordinates (default).

YES
Output modal coordinates to the text file **jobname.MCF**.

Damp

Damping Mode for Frequency Dependent Material Properties. This is only valid when using ANSYS DesignXplorer VT.

Hysteretic
Not proportional to the frequency.

Viscous
Proportional to the frequency (default).

Notes

Specifies the method of solution for a harmonic analysis (**ANTYPE**,HARMIC). If used in SOLUTION, this command is valid only within the first load step. See "Product Restrictions" below.

This command is also valid in PREP7.

Product Restrictions

For a harmonic analysis in ANSYS Professional, the default is *Method* = MSUP; *Method* = FULL or REDUC is not allowed.

For a harmonic analysis in Variational Technology, *MAXMODE* and *MINMODE* are not allowed, and *Method* = FULL, REDUC, and MSUP are not allowed.

Menu Paths

Main Menu>DesignXplorer VT>Solution>Solve
Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

HROUT, *Reimky*, *Clust*, *Mcont*

Specifies the harmonic analysis output options.

SOLUTION: Dynamic Options

MP ME ST <> <> PR EM <> <> PP ED

Reimky
Real/Imaginary print key:

ON
Print complex displacements as real and imaginary components (default).

OFF
Print complex displacements as amplitude and phase angle (degrees).

Clust
Cluster option (for **HROPT**,MSUP):

OFF
Uniform spacing of frequency solutions (default).

ON
Cluster frequency solutions about natural frequencies.

Mcont
Mode contributions key (for **HROPT**,MSUP):

OFF
No print of mode contributions at each frequency (default).

ON
Print mode contributions at each frequency.

Notes

Specifies the harmonic analysis (**ANTYPE**,HARMIC) output options. If used in SOLUTION, this command is valid only within the first load step. **OUTPR**,NSOL must be specified to print mode contributions at each frequency.

This command is ignored if the **HROPT** command has been issued with *Method* = SX or *Method* = SXRU, i.e., we do not have a printout at the expanded frequencies.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

I Commands

IC, *NODE*, *Lab*, *VALUE*, *VALUE2*, *NEND*, *NINC*
Specifies initial conditions at nodes.

SOLUTION: Misc Loads

MP ME ST <> <> PR EM <> FL PP ED

NODE
Node at which initial condition is to be specified. If ALL, apply to all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NODE*.

Lab
Degree of freedom label for which initial condition is to be specified. If ALL, use all appropriate labels. Structural labels: UX, UY, or UZ (displacements or linear velocities); ROTX, ROTY, or ROTZ (rotations or angular velocities). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). FLOTRAN fluid labels: PRES (pressure); VX, VY, or VZ (velocities); ENKE or ENDS (turbulent kinetic energy or turbulent energy dissipation); SP01 through SP06 (multiple species mass fractions) or their user-defined names [**MSSPEC**]. Electric label: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AX, AY, or AZ (vector magnetic potentials).

VALUE
Initial value of the degree of freedom (first-order value). Defaults to the program default for that DOF (0.0 for structural analysis, **TUNIF** for thermal analysis, etc.). Values are in the nodal coordinate system and in radians for rotational DOF.

VALUE2
Second-order degree of freedom value, mainly used to specify initial structural velocity. Defaults to the program default for that DOF (0.0 for structural analysis). Values are in the nodal coordinate system and in radians/time for rotational DOF.

NEND, *NINC*
Specifies the same initial condition values at the range of nodes from *NODE* to *NEND* (defaults to *NODE*), in steps of *NINC* (defaults to 1).

Notes

The **IC** command specifies initial conditions, which are the initial values of the specified degrees of freedom. It is valid only for a static analysis and the first load step of a full method transient analysis (**TIMINT**,ON and **TRNOPT**,FULL).

Initial conditions should always be step applied [**KBC**,1] and not ramped. (In a transient analysis when **SOLCONTROL** is ON, **KBC**,1 is the default as long as **TIMINT** is also on.)

If constraints [**D**, **DSYM**, etc.] and initial conditions are applied at the same node, the constraint specification will override.

For thermal analyses, any **TUNIF** specification should be specified before the **IC** command; otherwise, the **TUNIF** specification will be ignored. If the **IC** command is input before any **TUNIF** specification, use the **ICDELE** command and then reissue any **TUNIF** specification and then follow with the **IC** command.

Caution: Be careful not to define inconsistent initial conditions. For instance, if you define an initial velocity at a single DOF, the initial velocity at every other DOF will be 0.0, potentially leading to conflicting initial conditions. In most cases, you will want to define initial conditions at every unconstrained DOF in your model.

Once a solution has been performed, the specified initial conditions are overwritten by the actual solution and are no longer available. You must respecify them if you want to perform a re-analysis. You may want to keep a database file saved prior to the first solution or write to the **cdwrite** file (**CDWRITE**,LOAD) for subsequent reuse.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Initial Condit'n>Define
Main Menu>Solution>Define Loads>Apply>Initial Condit'n>Define

---

ICDELE
Deletes initial conditions at nodes.

SOLUTION: Misc Loads

MP ME ST <> <> PR EM <> FL PP ED

Notes

Deletes all initial conditions previously specified with the **IC** command at all nodes.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>Initial Condit'n
Main Menu>Solution>Define Loads>Delete>Initial Condit'n

ICE, *ELEM*, *Lab*, *VALUE*
Specifies initial conditions on elements.

SOLUTION: Misc Loads

MP ME ST <> <> PR EM <> FL PP ED

ELEM
Element to which initial condition applies. If All, apply initial condition to all selected elements [**ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

Lab
Valid initial conditions label. FLOTRAN fluid labels: VFRC (volume fraction).

VALUE
Initial value for the volume fraction.

Notes

The **ICE** command specifies initial conditions on selected elements.

Once a solution has been performed, the specified initial conditions will be overwritten by the actual solution, and are no longer available. You must respecify them if you want to perform a re-analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>On Elements
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>On Elements

---

ICEDELE, *ELEM*, *Lab*

Deletes initial conditions on elements.

SOLUTION: Misc Loads

MP ME ST <> <> PR EM <> FL PP ED

ELEM
Element at which initial conditions are to be deleted. If ALL, initial conditions at all selected elements [**ESEL**] are deleted. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

Lab
Valid initial conditions label. FLOTRAN fluid labels: VFRC (volume fraction).

Notes

Deletes all initial conditions previously specified with the **ICE** command at all elements.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Volume Fract>Init Loads>On Elements
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Volume Fract>Init Loads>On Elements

---

ICELIST, *ELEM*, *Lab*

Lists initial conditions on elements.

SOLUTION: Misc Loads

MP ME ST <> <> PR EM <> FL PP ED

ELEM

List initial condition for elements for *ELEM* (defaults to ALL). If *ELEM* = All, initial conditions for all selected elements [**ESEL**} are listed. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in GUI). A component name may be substituted for *ELEM*.

Lab

Valid initial conditions label. FLOTRAN fluid labels: VFRC (volume fraction).

Notes

Lists the initial on elements specified by the **ICE** command. Listing applies to all the selected elements [**ESEL**].

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Elem Init Condit'n>On Picked Elemts

ICLIST, *NODE1*, *NODE2*, *NINC*, *Lab*

Lists the initial conditions.

SOLUTION: Misc Loads

MP ME ST <> <> PR EM <> FL PP ED

NODE1, *NODE2*, *NINC*

List initial conditions for nodes *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL (default), *NODE2* and *NINC* are ignored and initial conditions for all selected nodes [**NSEL**] are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Lab

Velocity key:

DISP

Specification is for first order degree of freedom value (displacements, temperature, etc.) (default).

VELO

Specification is for second order degree of freedom value (velocities).

Notes

Lists the initial conditions specified by the **IC** command. Listing applies to all the selected nodes [**NSEL**] and DOF labels. **ICLIST** is not the same as the **DLIST** command. All the initial conditions including the default conditions are listed for the selected nodes.

Initial conditions for a FLOTRAN analysis are primary (first order) degrees of freedom and are thus listed with the DISP key.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Initial Condit'n>List Picked
Main Menu>Solution>Define Loads>Apply>Initial Condit'n>List Picked
Utility Menu>List>Loads>Initial Conditions>On Picked Nodes

/ICLWID, *FACTOR*

Scales the line width of circuit builder icons.

GRAPHICS: Scaling

MP <> <> <> <> <> EM <> <> PP ED

FACTOR
Multiplication factor applied to the default line width (defaults to 1). The minimum is 1 and the maximum is 6.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Scale Icon

/ICSCALE, *WN*, *FACTOR*

Scales the icon size for elements supported in the circuit builder.

GRAPHICS: Scaling

MP <> <> <> <> <> EM <> <> PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

FACTOR
Factor applied to the default icon size (defaults to 1).

Notes

Scaling the icon size can provide better visualization of the circuit components when using the Circuit Builder (an interactive builder available in the ANSYS GUI).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Scale Icon

ICVFRC, *Geom*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

Sets the initial volume fraction field for a geometry.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

Geom
Geometry:

CIRC
Circle.

ELPT
Ellipse.

VAL1, *VAL2*
Location of the center. *VAL1* and *VAL2* are the x and y coordinates, respectively.

VAL3, *VAL4*
If *Geom* = CIRC, *VAL3* is the radius of the circle and *VAL4* is not used. If *Geom* = ELPT, *VAL3* and *VAL4* are the x and y semiaxes of the ellipse, respectively.

Notes

The **ICVFRC** command sets the initial volume fraction field for a geometry. The initial VFRC is set to one for elements completely within the geometry. The initial VFRC is set equal to the fraction of element area within the geometry for elements intersected by the boundary of the geometry.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>By Geom>Circle
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>By Geom>Elliptic
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>By Geom>Circle
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Volume Fract>Init Loads>By Geom>Elliptic

---

IGESIN, *Fname*, *Ext*, --

Transfers IGES data from a file into ANSYS.

AUX15: IGES

MP ME ST DY <> PR EM EH FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to CAD if *Fname* is blank.

--

Unused field

Notes

Reads a file containing IGES data and transfers it into the ANSYS database. The file transferred is the IGES Version 5.1, ASCII format file. IGES (Initial Graphics Exchange Specification) is a neutral format developed by the U.S. Dept. of Commerce, National Institute of Standards and Technology. There is no output transfer file written since the transferred data is read directly into the ANSYS database.

You can import multiple files into a single database, but you must use the same import option (set with the **IOPTN** command) for each file.

The **IOPTN** command sets the parameters for reading the file (refer to this command for the options and their support for IGES entities). The two main parameters are FACETED and SMOOTH. You cannot change the **IOPTN** settings once a file has been imported.

When a file is read using the FACETED option, the data is stored in an defeaturing database. Once the transferred data is stored, a special set of geometry repair and enhancement tools, designed specifically for use with files imported from CAD systems, are available. Files read through the SMOOTH option use the standard database.

Menu Paths

Utility Menu>File>Import

IGESOUT, *Fname*, *Ext*, *--*, *ATT*

Writes solid model data to a file in IGES Version 5.1 format.

PREP7: Database

MP ME ST DY <> PR EM EH FL PP <>

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext

Filename extension (8 character maximum).

The extension defaults to IGES if *Fname* is blank.

--

Unused field

ATT

Attribute key:

0

Do not write assigned numbers and attributes of the solid model entities to the IGES file (default).

1
Write assigned numbers and attributes of solid model entities (keypoints, lines, areas, volumes) to the IGES file. Attributes include MAT, TYPE, REAL, and ESYS specifications as well as associated solid model loads and meshing (keypoint element size, number of line divisions and spacing ratio) specifications.

Notes

Causes the selected solid model data to be written to a coded file in the IGES Version 5.1 format. Previous data on this file, if any, are overwritten. Keypoints that are not attached to any line are written to the output file as IGES entity 116 (Point). Lines that are not attached to any area are written to the output file as either IGES Entity 100 (Circular Arc), 110 (Line), or 126 (Rational B-Spline Curve) depending upon whether the ANSYS entity was defined as an arc, straight line, or spline. Areas are written to the output file as IGES Entity 144 (Trimmed Parametric Surface). Volumes are written to the output file as IGES entity 186 (Manifold Solid B-Rep Object). Solid model entities to be written must have all corresponding lower level entities selected (use **ALLSEL**,BELOW,ALL) before issuing command. Concatenated lines and areas are not written to the IGES file; however, the entities that make up these concatenated entities are written.

Caution: If you issue the **IGESOUT** command after generating a beam mesh with orientation nodes, the orientation keypoints that were specified for the line [**LATT**] are no longer associated with the line and are not written out to the IGES file. The line does not recognize that orientation keypoints were ever assigned to it, and the orientation keypoints do not "know" that they are orientation keypoints. Thus the **IGESOUT** command does not support (for beam meshing) any line operation that relies on solid model associativity. For example, meshing the areas adjacent to the meshed line, plotting the line that contains the orientation nodes, or clearing the mesh from the line that contains orientation nodes may not work as expected. See Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide* for more information about beam meshing.

You cannot access this command for models that have been imported from IGES using the FACETED translator (see the **IOPTN** command).

Menu Paths

Utility Menu>File>Export

/IMAGE, *Label*, *Fname*, *Ext*, --

Allows graphics data to be captured and saved.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Lab
Label specifying the operation to be performed:

CAPTURE
Capture the image from the graphics window to a new window.

RESTORE
Restore the image from a file to a new window.

SAVE
Save the contents of the graphic window to a file.

DELETE
Delete the window that contains the file.

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Ext
Filename extension (8 character maximum).

If no extension is specified, **bmp** will be used on Windows systems, and **img** will be used on UNIX systems.

--
Unused field

Menu Paths

This command cannot be accessed from a menu.

IMAGIN, *IR*, *IA*, *--*, *--*, *Name*, *--*, *--*, *FACTA*

Forms an imaginary variable from a complex variable.

POST26: Operations

MP ME ST DY <> PR EM <> <> PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA
Reference number of the variable to be operated on.

--, *--*
Unused fields.

Name
Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, *--*
Unused fields.

FACTA
Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

Notes

This command forms a new variable from a complex variable by storing the imaginary part as the real part. The imaginary part can then be used in other operations. Used only with harmonic analyses (**ANTYPE**,HARMIC).

Complex variables are stored in two-column arrays with the real component stored in the first column and the imaginary component stored in the second column. This command extracts the value stored in the second column (i.e., imaginary component). However, with harmonic analyses, all variables are stored in two-column arrays as complex variables. If the variable is not complex, then the same value is stored in both columns. This

command will extract the variable in the second column of the array, even if this variable is not the imaginary component of a complex variable.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Imaginary Part

IMESH, *LAKY*, *NSLA*, *NTLA*, *KCN*, *DX*, *DY*, *DZ*, *TOL*

Generates nodes and interface elements along lines or areas.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

LAKY
Copies mesh according to the following:

LINE or 1
Copies line mesh (default).

AREA or 2
Copies area mesh.

NSLA
Number that identifies the source line or area. This is the line or area whose mesh will provide the pattern for the interface elements. ANSYS copies the pattern of the line or area elements through the area or volume to create the mesh of area or volume interface elements.

NTLA
Number that identifies the target line or area. This is the line or area that is opposite the source line or area specified by *NSLA*. Add *NTLA* to obtain the copied mesh from the source line or area.

KCN
Number that identifies the particular ANSYS coordinate system.

DX, *DY*, *DZ*
Incremental translation of node coordinates in the active coordinate system (*DR*, *D* , *DZ* for cylindrical, and *DR*, *D* , *D* for spherical or toroidal). The source line or area coordinates + *DX*, *DY*, *DZ* = the target line or area coordinates. If left blank, ANSYS automatically estimates the incremental translation.

TOL
Tolerance for verifying topology and geometry. By default, ANSYS automatically calculates the tolerance based on associated geometries.

Notes

Generates nodes and interface elements along lines or areas. The **IMESH** command requires that the target line or area exactly match the source line or area. Also, both target and source lines or areas must be in the same area or volume. The area or volume containing the source line or area must be meshed before executing **IMESH**, while the area or volume containing the target line or area must be meshed after executing **IMESH**.

For three dimensional problems where *LAKY* = AREA, ANSYS fills the interface layer according to the following table:

If source mesh consists of:	ANSYS fills the interface layer with:
Quadrilateral elements	Hexahedral elements
Triangle elements	Degenerated wedge elements
Combination quadrilateral and triangle elements	Combination hexahedral and degenerated wedge elements

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Interface Mesh>2D Interface
Main Menu>Preprocessor>Meshing>Mesh>Interface Mesh>3D Interface

---

IMMED, *KEY*

Allows immediate display of a model as it is generated.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

KEY

Immediate mode key:

0

Display only upon request, i.e., no immediate display (default with the GUI *off*).

1

Display immediately as model is generated (default with the GUI *on*).

Command Default

As described above.

Notes

Allows immediate display of a model (as it is generated) without a screen erase or a display request. Available only during an interactive session at a graphics display terminal. A valid graphics device name must first be specified on the **/SHOW** command.

The **IMMED** command allows you to control whether or not the model is displayed immediately as it is generated in an interactive session. By default in the GUI, your model will immediately be displayed in the Graphics Window as you create new entities (such as areas, keypoints, nodes, elements, local coordinate systems, boundary conditions, etc.). This is called *immediate mode* graphics. Also note that symbols (such as boundary conditions, local coordinate system triads, etc.) are shown immediately and will be present on subsequent displays unless you "turn off" the appropriate symbol using the GUI plot controls function or the appropriate graphics specification command.

An immediate image will also be automatically scaled to fit nicely within the Graphics Window -- a feature called *automatic scaling*. The new scaling is usually apparent on the automatic replot associated with immediate mode. To suppress automatic replot, issue **/UIS**,REPLOT,0. (With automatic replot suppressed, the immediate image may not always be automatically scaled correctly.)

Note — An immediate display in progress should *not* be aborted with the usual system "break" feature (or else the ANSYS session itself will be aborted). When you run the ANSYS program interactively *without* using the GUI, immediate mode is off by default.

This command is valid only in PREP7.

Menu Paths

Utility Menu>PlotCtrls>Erase Options>Immediate Display

IMPD, *Vpath*, *Ipath*, *Vsymm*, *Isymm*

Calculates the impedance of a conductor at a reference plane.

POST1: Magnetics Calculations

MP ME <> <> <> <> EM <> <> PP ED

Vpath
Path name for a predefined path [**PATH** command] for calculating the EMF (voltage drop) from the conductor to a reference point. The path should start at the outer conductor wall and end at a reference voltage point.

Ipath
Path name for a predefined path [**PATH** command] for calculating the MMF (current) in a conductor. The path should traverse a closed contour surrounding the conductor, and you should define the path in a counterclockwise direction.

Vsymm
Symmetry factor applied to the calculated EMF (voltage drop). The EMF (voltage drop) from the conductor to the reference point is multiplied by *Vsymm*.

Isymm
Symmetry factor applied to the calculated current. The calculated current is multiplied by *Isymm*.

Notes

Used in a harmonic high-frequency electromagnetic analysis, **IMPD** calculates the impedance of a conductor at a reference plane from the EMF (voltage) and MMF (current) at the reference plane. The EMF (voltage drop) is calculated by a line integral from the input path name (specified by the *Vpath* argument) that extends from the conductor to a reference point. The MMF (current) is calculated by a closed path around the conductor from the input path name (specified with *Ipath*). In cases having modeled symmetry, you can multiply the voltage drop or current by symmetry factors (*Vsymm* and *Isymm* respectively).

This command macro returns the scalar parameters Zre and Zim, representing the real and imaginary components of the impedance.

See magnetic macros for further details.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>Impedance

---

/INPUT, *Fname*, *Ext*, --, *LINE*, *LOG*

Switches the input file for the commands that follow.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to the current **Jobname** if *Ext* is specified.

Ext
Filename extension (8 character maximum).

--
Unused field

LINE
A value indicating either a line number in the file or a user-defined label in the file from which to begin reading the input file.

(blank), 0, or 1
Begins reading from the top of the file (default).

LINE_NUMBER
Begins reading from the specified line number in the file.

:*label*
Begins reading from the first line beginning with the matching user-defined label :*label* (beginning with a colon (:), 8 characters maximum).

LOG
Indicates whether secondary input from this command should be recorded in the command log (**File.LOG**) and the database log:

0
Record only the **/INPUT** command on the log (default).

1
Record commands in the specified secondary file as they are executed.

Notes

Switches the input file for the next commands. Commands are read from this file until an end-of-file or another file switching directive is read. An end-of-file occurs after the last record of the file or when a **/EOF** command is read. An automatic switch back one level (to the previous file) occurs when an end-of-file is encountered. Twenty levels of nested file switching are allowed. Note that files including ***DO**, ***USE**, ***ULIB**, and the "Unknown Command" Macro have less nesting available because each of these operations also uses a level of file switching. For an interactive run, a **/INPUT**,TERM switches to the terminal for the next input. A **/EOF** read from the terminal then switches back to the previous file. A **/INPUT** (with a blank second field) switches back to the primary input file.

Setting *LOG* = 1 on **/INPUT** causes all commands read from the specified file to be recorded in the command log (**File.LOG**) and the internal database command log [**LGWRITE**]. This option is recommended if the log file will be used later (e.g., as batch input or as an analysis file for design optimization). The *LOG* = 1 option is only valid

when the **/INPUT** occurs in the primary input file. Using *LOG* = 1 on a nested **/INPUT** or on a **/INPUT** within a do-loop will have no effect (i.e., commands in the secondary input file are not written to the command log).

This command is valid in any processor.

Menu Paths

Utility Menu>File>Read Input from

INRES, *Item1*, *Item2*, *Item3*, *Item4*, *Item5*, *Item6*, *Item7*, *Item8*

Identifies the data to be retrieved from the results file.

POST1: Controls

MP ME ST DY <> PR EM <> FL PP ED

Item1, *Item2*, *Item3*, *Item4*, *Item5*, *Item6*, *Item7*, *Item8*
Data to be read into the database from the results file. May consist of any of the following labels:

ALL
All solution items (default).

BASIC
NSOL, RSOL, NLOAD, STRS, FGRAD, and FFLUX items.

NSOL
Nodal DOF solution.

RSOL
Nodal reaction loads.

ESOL
Element solution items (includes all of the following):

NLOAD
Element nodal loads.

STRS
Element nodal stresses.

EPEL
Element elastic strains.

EPTH
Element thermal, initial, and swelling strains.

EPPL
Element plastic strains.

EPCR
Element creep strains.

FGRAD
Element nodal gradients.

FFLUX
Element nodal fluxes.

MISC
Element miscellaneous data (SMISC and NMISC).

Notes

Identifies the type of data to be retrieved from the results file for placement into the database through commands such as **SET**, **SUBSET**, and **APPEND**. **INRES** is a companion command to the **OUTRES** command controlling data written to the database and the results file. Since the **INRES** command can only flag data that has already been written to the results file, care should be taken when using the **OUTRES** command to include all data you wish to retrieve for postprocessing later on.

Menu Paths

Main Menu>General Postproc>Data & File Opts

INRTIA
Specifies "Inertial loads" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Loads>Inertia Loads
Utility Menu>List>Status>Solution>Inertia Loads

INT1, *IR*, *IY*, *IX*, --, *Name*, --, --, *FACTA*, *FACTB*, *CONST*
Integrates a variable.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result. Table values represent integrated sum of *IY* to current table position of *IX*.

IY, *IX*
Integrate variable *IY* with respect to *IX*.

--
Unused field.

Name
Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --
Unused fields.

FACTA, FACTB
Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

CONST
Initial value.

Notes

Integrates variables according to the operation:

$$IR = \int (FACTA \times IY)\, d(FACTB \times IX) + CONST$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Integrate

INTSRF, *Lab*

Integrates nodal results on an exterior surface.

POST1: Special Purpose

MP ME ST DY <> <> <> <> FL PP ED

Lab
Label indicating degree of freedom to be integrated:

PRES
Pressure.

TAUW
Wall shear stress.

FLOW
Both pressure and wall shear stress.

Notes

Integrates nodal results on a surface. Use node selection (such as the EXT option of the **NSEL** command) to indicate the surface(s) of element faces to be used in the integration. A surface can be "created" by unselecting elements (such as unselecting non-fluid elements that are adjacent to fluid elements for the postprocessing of fluid flow result). Element faces attached to the selected nodes will be automatically determined. All nodes on a face must be selected for the face to be used. The integration results will cancel for nodes on common faces of adjacent selected elements.

Integration results are in the active coordinate system (see the **RSYS** command). The type of results coordinate system must match the type used in the analysis. However, you may translate and rotate forces and moments as needed. Use the ***GET** command (**Utility Menu> Parameters> Get Scalar Data**) to retrieve the results.

Menu Paths

Main Menu>General Postproc>Nodal Calcs>Surface Integral

IOPTN, *Lab*, *VAL1*

Controls options relating to importing a model.

AUX15: IGES

MP ME ST DY <> PR EM <> FL PP ED

Lab

Label identifying the import option. The meaning of *VAL1* will vary depending on *Lab*.

STAT
: List overall status of import facilities, including current option values. *VAL1* is ignored.

DEFA
: Set default values for all import options. *VAL1*is ignored.

MERG
: Entity merge option. *VAL1* can be:

  YES
  : Automatic merging is performed (default).

  NO
  : No merging of entities.

SOLID
: Solid option. *VAL1* can be:

  YES
  : Solid is created automatically (default).

  NO
  : No solid created.

GTOLER
: Entity merging tolerance. Valid arguments for *VAL1* depend on value of IGES. If IGES = FACETED, then GTOLER,*VAL1* is a numeric value used as a multiplying factor to change the maximum model dimension. If IGES = SMOOTH, then GTOLER,*VAL1* can be:

  DEFA
  : Use system defaults (default).

  FILE
  : Use tolerance from the imported file.

  n
  : A user-specified tolerance value.

IGES
: IGES import option. *VAL1* can be:

STAT
List status of IGES related options in the output window.

SMOOTH (or RV52)
Use more robust IGES revision 5.2 import function (default).

FACETED (or RV53)
Use defeaturing database.

SMALL
Small areas option. *VAL1* can be:

YES
Small areas are deleted (default).

NO
Small areas are retained.

VAL1
Additional input value as described under each option for *Lab*.

Command Default

Merging will be performed during the IGES transfer with no global solid model tolerance (GTOLER) used.

Notes

Controls various options during a model file transfer (e.g., **IGESIN** command). A global solid model tolerance (GTOLER) can be specified.

IOPTN,SMALL,YES (default value) will delete small areas and can cause geometrical inconsistencies that could cause the import process to abort. Retaining the small areas will increase processor time and memory usage.

The two IGES translation options, SMOOTH and FACETED, determine which graphics database will be used when IGES files are read into ANSYS. If the SMOOTH option is chosen (default), the data is stored in the standard ANSYS graphics database. If the FACETED option is used, the defeaturing database is used and a special set of geometry repair and enhancement tools, designed specifically for use with files imported from CAD systems, are made available.

The SMOOTH option is capable of reading in any rational B-spline curve entity (type 126), or rational B-spline surface entity (type 128) with a degree less than or equal to 20. Attempts to read in B-spline curve or surface entities of degree higher than 20 may result in error messages.

The FACETED option translates all IGES topological and geometric entities. The option ignores data such as dimensions, text, annotation entities, structure entities, and any IGES entities that the filter doesn't recognize. The FACETED option is not recommended for large, complex geometries.

If you issue the **/CLEAR** command, the **IOPTN** settings will return to their default.

If **IOPTN**, MERG, YES is used, merging of coincident geometry items is performed automatically when the **IGESIN** command is issued. (In other words, an internal **NUMMRG**,KP command is issued.) The model will be merged with the *consideration tolerance* (*T0LER* on **NUMMRG**) set equal to 0.75 * the shortest distance between the endpoints of any active line. See the **NUMMRG** command for more information on these tolerances. In most cases, this default merging is appropriate. The **IOPTN** command should be used when you want to turn off merging

operations, override the default merging and specify a global solid model tolerance value (GTOLER), or turn off the automatic creation of solids (SOLID). If used, the **IOPTN** command should be issued *before* the **IGESIN** command. You cannot change these options once your model has been imported or created. If you must change these options, clear the database using the **/CLEAR** command (or exit and restart ANSYS), set the correct options, and reimport or recreate the model.

This command is valid in any processor.

Menu Paths

Utility Menu>File>Import

IRLF, *KEY*

Specifies that inertia relief calculations are to be performed.

SOLUTION: Inertia

MP ME ST <> <> PR <> <> <> PP ED

KEY

Calculation key:

0
: No inertia relief calculations.

1
: Counterbalance loads with inertia relief forces.

1
: Precalculate masses for summary printout only (no inertia relief).

Command Default

No inertia relief calculations.

Notes

Specifies that the program is to calculate accelerations to counterbalance the applied loads (inertia relief). Displacement constraints on the structure should be only those necessary to prevent rigid-body motions (3 are needed for a 2-D structure and 6 for a 3-D structure). The sum of the reaction forces at the constraint points will be zero. Accelerations are calculated from the element mass matrices and the applied forces. Data needed to calculate the mass (such as density) must be input. Both translational and rotational accelerations may be calculated.

This option applies only to the static (**ANTYPE**,STATIC) analysis. Nonlinearities, substructures, elements that operate in the nodal coordinate system, and axisymmetric elements are not allowed. Models with both 2-D and 3-D element types are not recommended. Loads may be input as usual. Displacements and stresses are calculated as usual. Use **IRLIST** to print inertia relief calculation results. The mass and moment of inertia summary printed before the solution is accurate (because of the additional pre-calculations required for inertia relief). See the *ANSYS, Inc. Theory Reference* for calculation details. See also the *ANSYS Structural Analysis Guide* for procedural details.

If the inertia relief calculation is to be performed in the second or later load step, you must specify **EMATWRITE**,YES in the initial load step for the element matrices needed to perform the calculations to be available.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Inertia Relief
Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Incl Mass Summry
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Inertia Relief
Main Menu>Solution>Load Step Opts>Output Ctrls>Incl Mass Summry

---

IRLIST
Prints inertia relief summary table.

POST1: Listing

MP ME ST DY <> PR <> <> <> PP ED

Notes

Prints the inertia relief summary data, including the mass summary table, the total load summary table, and the inertia relief summary table resulting from the inertia relief calculations. These calculations are performed in the solution phase [**SOLVE** or **PSOLVE**] as specified by the **IRLF** command.

Inertia relief output is stored in the database rather than in the results file (**Jobname.RST**). When you issue **IRLIST**, ANSYS pulls the information from the database, which contains the inertia relief output from the most recent solution [**SOLVE** or **PSOLVE**].

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

ISFILE, *Option*, *Fname*, *Ext*, *--*, *LOC*, *MAT1*, *MAT2*, *MAT3*, *MAT4*, *MAT5*, *MAT6*, *MAT7*, *MAT8*, *MAT9*, *MAT10*
Reads an initial stress state from a file into ANSYS.

SOLUTION: Misc Loads

MP ME ST <> <> PR <> <> <> PP ED

Option
Label identifying the option to be performed.

READ
Read initial stress data from the specified file (default).

LIST
List initial stresses that have been previously read in. *LIST* may take an elements ID or ALL as an option to list a particular layer of the element.

DELE
Delete initial stresses that have been previously read in. *DELE* may take an elements ID or ALL as an option to list a particular layer of the element.

Fname
If *Option* = READ, *Fname* is the file name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name. Defaults to **Jobname**.

If *Option* = LIST or DELETE, input the element number for which initial stresses are to be listed or deleted in this field. If this field is blank, it defaults to all currently selected elements. For the layered SHELL181, SHELL208, and SHELL209 elements, the next field (*Ext*) can be used to list or delete the initial stress specification for a specific layer of the element. Remaining fields are ignored.

Ext
Filename extension (8 character maximum).

The extension defaults to IST if *Fname* is blank.

Layer number if *Option* = LIST or DELETE and the element type is layered SHELL181, SHELL208, and SHELL209.

--
Unused field

LOC
Global location flag. This flag indicates where the initial stresses are to be applied within each element.

0
Element centroid (default).

1
Integration points.

2
Element specific locations. For this option, you must specify a location flag with each element stress record in the initial stress file.

3
Constant stress state. The first stress record on the initial stress file is used to initiate a constant stress state for all elements.

MAT1, MAT2, MAT3, MAT4, MAT5, MAT6, MAT7, MAT8, MAT9, MAT10
Materials to which the initial stresses should apply. If not specified, the stresses apply to all materials.

Notes

This command reads initial stresses from an ASCII file (initial stress file) and applies them as loads in an ANSYS structural analysis. The initial stress import capability is supported by the following element types: PLANE2, PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209. This command operates only on the currently selected set of elements. The element numbers in the initial stress file must match those in the selected set. The initial stresses must be specified in the element local coordinate system. If an element coordinate system (ESYS) is defined for an element, the initial stresses must be specified in this coordinate system. The format of the initial stress file is documented in Initial Stress Loading in the *ANSYS Basic Analysis Guide*.

The optional list of materials can be specified as individual material numbers and/or ranges of materials. A range is specified by using three consecutive *MAT* slots with the first slot being one end of the range, the second slot being the other end of the range, and the third slot being the increment to use for the range. It is not necessary to enter the lower end of the range first. The increment must be entered as a negative number. For example, to specify that the stresses apply to material 1 and the even material numbers from 2 to 8, you would enter 1,2,8,-2 in four consecutive *MAT* slots.

The option *READ* allows for layered SHELL181, SHELL208, and SHELL209 elements. The element record on the initial stress file is automatically extended to allow for initial stress specification for every layer.

The options *LIST* and *DELE* can take an element's ID or ALL to list a particular layer of the element. If the element number is blank, it lists or deletes the initial stress definition for all elements. If the specific element number is defined, it does the listing or deleting for all layers of the element. If a layer number is provided, it does those operations only for that layer. The initial stress command will overwrite any previous initial stress specification.

You cannot use more than one method (**ISTRESS**, **ISFILE**, or the USTRESS user subroutine) to input initial stresses for an element. **ISWRITE** can be used to write out initial stress values to an ASCII file.

The **ISFILE** command will overwrite any previous initial stress specification applied through the **ISFILE** or **ISTRESS** commands.

The **ISFILE** command is only available in SOLUTION. This command is not supported by **CDWRITE**.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Initial Stress>Delete Stresses
Main Menu>Preprocessor>Loads>Load Step Opts>Initial Stress>List Stresses
Main Menu>Preprocessor>Loads>Load Step Opts>Initial Stress>Read IS File
Main Menu>Solution>Load Step Opts>Initial Stress>Delete Stresses
Main Menu>Solution>Load Step Opts>Initial Stress>List Stresses
Main Menu>Solution>Load Step Opts>Initial Stress>Read IS File

---

ISTRESS, *Sx*, *Sy*, *Sz*, *Sxy*, *Syz*, *Sxz*, *MAT1*, *MAT2*, *MAT3*, *MAT4*, *MAT5*, *MAT6*, *MAT7*, *MAT8*, *MAT9*, *MAT10*
Defines a set of initial stress values.

SOLUTION: Misc Loads

MP ME ST DY <> PR EM <> FL PP ED

Sx, Sy, Sz, Sxy, Syz, Sxz
Initial, constant stress values.

MAT1, MAT2, MAT3, MAT4, MAT5, MAT6, MAT7, MAT8, MAT9, MAT10
Materials to which the initial stresses should apply. If not specified, the stresses apply to all materials.

Notes

ISTRESS is available only in the solution phase. The command provides a set of initial stress values which are applied in an ANSYS structural analysis. It operates only on the currently selected set of elements, and is supported by the following element types: PLANE2, PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209.

The initial stresses must be specified in the element local coordinate system. If an element coordinate system (**ESYS** command) is defined for an element, the initial stress values must be supplied in that coordinate system. After setting the stress, it may be listed or deleted for any specific element or for all elements using **ISFILE**.

The optional list of materials can be specified as individual material numbers and/or ranges of materials. A range is specified by using three consecutive *MAT* slots with the first slot being one end of the range, the second slot being the other end of the range, and the third slot being the increment to use for the range. It is not necessary to enter the lower end of the range first. The increment must be entered as a negative number. For example, to specify that the stresses apply to material 1 and the even material numbers from 2 to 8, you would enter 1,2,8,-2 in four consecutive *MAT* slots.

You cannot use more than one method (**ISTRESS**, **ISFILE**, or the USTRESS user subroutine) to input initial stresses for an element. The **ISTRESS** command will overwrite the initial stress specified previously by the **ISFILE** or **ISTRESS** commands. This command is not supported by the **CDWRITE** command.

A static analysis with initial stress cannot be used in a subsequent prestressed modal or buckling analysis.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Initial Stress>Apply Const Strs
Main Menu>Solution>Load Step Opts>Initial Stress>Apply Const Strs

---

ISWRITE,*Switch*

Writes an ASCII file containing the initial stress values.

SOLUTION: Misc Loads

MP ME ST DY <> PR EM <> FL PP ED

Switch
Determines whether an initial stress file is written (or not).

ON
Writes an initial stress file with the name **Jobname.ist**.

OFF
Does not write the initial stress file.

Notes

ISWRITE is available only in the solution phase. The command writes a file containing the stresses. Should an initial stress file of the same name already exist, the file is overwritten.

This command is not supported by the **CDWRITE** command.

For nonlinear analysis, the stresses are calculated at the integration points when convergence occurs. For linear analysis, the stresses are calculated when the solution is finished.

The format of the initial stress file is documented in Initial Stress Loading in the *ANSYS Basic Analysis Guide*

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Initial Stress>Write Stresses

Main Menu>Solution>Load Step Opts>Initial Stress>Write Stresses

J Commands

JPEG, *Kywrd*, *OPT*

Provides JPEG file export for ANSYS displays.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Kywrd

Specifies various JPEG file export options.

QUAL

If *Kywrd* = QUAL, then *OPT* is an integer value defining the JPEG quality index on an arbitrary scale ranging from 1 to 100. The default value is 75.

ORIENT

If *Kywrd* = ORIENT, then *OPT* will determine the orientation of the entire plot. *OPT* can be either Horizontal (default) or Vertical.

COLOR

If *Kywrd* = COLOR, then *OPT* will determine the color depth of the saved file. *OPT* can be 0, 1, or 2, corresponding to Black and White, Grayscale, and Color (default), respectively.

TMOD

If *Kywrd* = TMOD, then *OPT* will determine the text method. *OPT* can be either 1 or 0, corresponding to bitmap text (default) or line stroke text, respectively.

DEFAULT

If *Kywrd* = DEFAULT, then all of the default values, for all of the Kywrd parameters listed above, are active.

OPT

OPT can have the following names or values, depending on the value for *Kywrd* (see above).

1 to 100

If *Kywrd* = QUAL, a value between 1 and 100 will determine the quality index of the JPEG file.

Horizontal, Vertical

If *Kywrd* = ORIENT, the terms Horizontal or Vertical determine the orientation of the plot.

0,1,2

If *Kywrd* = COLOR, the numbers 0, 1, and 2 correspond to Black and White, Grayscale and Color, respectively.

1,0

If *Kywrd* = TMOD, the values 1 and 0 determine whether bitmap (1) or stroke text (0) fonts will be used

Menu Paths

Utility Menu>PlotCtrls>HardCopy>ToFile

JSOL, *NVAR*, *ELEM*, *ITEM*, *COMP*, *Name*

Stores the relative displacement, relative rotation, reaction forces, and moments for the joint element.

POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

NVAR
Arbitrary reference number or name assigned to this variable. Variable numbers can be 2 to *NV* (**NUMVAR**) while the name can be an eight-byte character string. Overwrites any existing results for this variable.

ELEM
Element number for which to store results.

Item
Label identifying the item. Valid item labels are shown in JSOL - Valid Item and Component Labels below.

Comp
Component of the *Item* (if required). Valid component labels are shown in JSOL - Valid Item and Component Labels below.

Name
Thirty-two character name identifying the item on printouts and displays. Defaults to a label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Notes

Valid for the MPC184 joint element using KEYOPT(1) = 6 or 7.

JSOL - Valid Item and Component Labels

Item	Comp	Description
U	X, Y, Z	x, y, or z relative displacement.
ROT	X, Y, Z	x, y, or z relative rotation.
RF	X, Y, Z	Reaction forces in the local x, y, or z direction.
RM	X, Y, Z	Reaction moments in the local x, y, or z direction.

Menu Paths

This command cannot be accessed from a menu.

K Commands

K, *NPT*, *X*, *Y*, *Z*
Defines a keypoint.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

NPT
Reference number for keypoint. If zero, the lowest available number is assigned [**NUMSTR**].

X, *Y*, *Z*
Keypoint location in the active coordinate system (may be R, θ, Z or R, θ, Φ). If *X* = P, graphical picking is enabled and all other fields (including *NPT*) are ignored (valid only in the GUI).

Notes

Defines a keypoint in the active coordinate system [**CSYS**] for line, area, and volume descriptions. A previously defined keypoint of the same number will be redefined. Keypoints may be redefined only if it is not yet attached to a line or is not yet meshed. Solid modeling in a toroidal system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>In Active CS
Main Menu>Preprocessor>Modeling>Create>Keypoints>On Working Plane

---

KATT, *MAT*, *REAL*, *TYPE*, *ESYS*
Associates attributes with the selected, unmeshed keypoints.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

MAT, *REAL*, *TYPE*, *ESYS*
Material number, real constant set number, type number, and coordinate system number to be associated with selected, unmeshed keypoints.

Notes

Keypoints subsequently generated from the keypoints will also have these attributes. These element attributes will be used when the keypoints are meshed. If a keypoint does not have attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT**, **REAL**,**TYPE**, and **ESYS** command settings. Reissue the **KATT** command (before keypoints are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association.

If any of the arguments *MAT*, *REAL*, *TYPE*, or *ESYS* are defined as -1, then that value will be left unchanged in the selected set.

In some cases, ANSYS can proceed with a keypoint meshing operation even when no logical element type has been assigned via **KATT**,,,*TYPE* or **TYPE**. For more information, see the discussion on setting element attributes in Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>All Keypoints
Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked KPs

KBC, *KEY*

Specifies stepped or ramped loading within a load step.

SOLUTION: Load Step Options

MP ME ST <> <> PR EM <> <> PP ED

KEY

Ramping key:

0

Loads are linearly interpolated (ramped) for each substep from the values of the previous load step to the values of this load step.

1

Loads are step changed (stepped) at the first substep of this load step to the values of this load step (i.e., the same values are used for all substeps). Useful for rate-dependent behavior (e.g., creep, viscoplasticity, etc.) or transient load steps only.

Command Default

When **SOLCONTROL** is ON, ANSYS performs ramped loading if **ANTYPE**,STATIC, or if **ANTYPE**,TRANS and **TIMINT**,OFF. It performs stepped loading if **ANTYPE**,TRANS and **TIMINT**,ON. (In a transient analysis, **TIMINT**,ON is the default.)

When **SOLCONTROL** is OFF, ramped loading for all types of transient or nonlinear analysis.

Notes

Specifies whether loads applied to intermediate substeps within the load step are to be stepped or ramped. Used only if *DTIME* on the **DELTIM** command is less than the time span or, conversely, if *NSBSTP* on the **NSUBST** command is greater than one. Flags (FSI, MXWF, MVDI, etc.) are always stepped.

For ramped loading (**KBC**,0), when a load is applied for the first time, it is interpolated from zero to the value of the current load step, and not from the initial condition or value of the DOF from the previous load step. Tabular boundary conditions do not support ramping and instead apply their full value regardless of the **KBC** setting.

Irrespective of the **KBC** setting, loads are usually step-removed. See Stepping or Ramping Loads in the *ANSYS Basic Analysis Guide* for more information.

It sometimes is difficult to obtain successful convergence with stepped loading in a nonlinear transient problem. If divergence is encountered, check if stepped loading was used by default, then determine if it is appropriate for the analysis.

If you run an analysis with **SOLCONTROL**,ON, but do not issue the **KBC** command, ANSYS will choose whether or not to use stepped or ramped loads. The program-chosen option will be recorded on the load step files as **KBC**,-1.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Freq and Substps
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substps
Main Menu>Solution>Analysis Type>Sol'n Controls>Transient
Main Menu>Solution>Load Step Opts>Time/Frequenc>Freq and Substps
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substps

---

KBETW, *KP1*, *KP2*, *KPNEW*, *Type*, *VALUE*

Creates a keypoint between two existing keypoints.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

KP1
First keypoint. If *KP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

KP2
Second keypoint.

KPNEW
Number assigned to the new keypoint. Defaults to the lowest available keypoint number.

Type
Type of input for *VALUE*.

RATIO
Value is the ratio of the distances between keypoints as follows: (*KP1*-*KPNEW*)/(*KP1*-*KP2*).

DIST
Value is the absolute distance between *KP1* and *KPNEW* (valid only if current coordinate system is Cartesian).

VALUE
Location of new keypoint, as defined by *Type* (defaults to 0.5). If *VALUE* is a ratio (*Type* = RATIO) and is less than 0 or greater than 1, the keypoint is created on the extended line. Similarly, if *VALUE* is a distance (*Type* = DIST) and is less than 0 or greater than the distance between *KP1* and *KP2*, the keypoint is created on the extended line.

Notes

Placement of the new keypoint depends on the currently active coordinate system [**CSYS**]. If the coordinate system is Cartesian, the keypoint will lie on a straight line between *KP1* and *KP2*. If the system is not Cartesian (e.g., cylindrical, spherical, etc.), the keypoint will be located as if on a line (which may not be straight) created in the current coordinate system between *KP1* and *KP2*. Note that solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>KP between KPs

KCALC, *KPLAN*, *MAT*, *KCSYM*, *KLOCPR*

Calculates stress intensity factors in fracture mechanics analyses.

POST1: Special Purpose

MP ME ST <> <> PR <> <> <> PP ED

KPLAN

Plane stress key:

0
: Use plane strain or axisymmetric displacement extrapolation.

1
: Use plane stress displacement extrapolation.

MAT

Material number used in the extrapolation (defaults to 1).

KCSYM

Symmetry key:

0 or 1
: Half-crack model with symmetry boundary conditions [**DSYM**] in the crack-tip coordinate system. $K_{II} = K_{III} = 0$. Three nodes are required on the path.

2
: Like 1 except with antisymmetric boundary conditions ($K_I = 0$).

3
: Full-crack model (both faces). Five nodes are required on the path (one at the tip and two on each face).

KLOCPR

Local displacements print key:

0
: Do not print local crack-tip displacements.

1
: Print local displacements used in the extrapolation technique.

Notes

Calculates the stress intensity factors (K_I, K_{II}, and K_{III}) associated with homogeneous isotropic linear elastic fracture mechanics. A displacement extrapolation method is used in the calculation (see the *ANSYS, Inc. Theory Reference*). The **PATH** and **PPATH** commands must be used to define a path with the crack face nodes (*NODE1* at the crack tip, *NODE2* and *NODE3* on one face, *NODE4* and *NODE5* on the other (optional) face). A crack-tip coordinate system, having x parallel to the crack face (and perpendicular to the crack front) and y perpendicular to the crack face, must be the active RSYS and CSYS before **KCALC** is issued.

Menu Paths

Main Menu>General Postproc>Nodal Calcs>Stress Int Factr

KCENTER, *Type*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *KPNEW*

Creates a keypoint at the center of a circular arc defined by three locations.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

Type
Type of entity used to define the circular arc. The meaning of *VAL1-VAL4* will vary depending on *Type*. If *Type* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

KP
Arc is defined by keypoints.

LINE
Arc is defined by locations on a line.

VAL1, *VAL2*, *VAL3*, *VAL4*
Values used to specify three locations on the arc (see table below).

KPNEW
Number assigned to new keypoint. Defaults to the lowest available keypoint number.

VAL1-VAL4 Definitions

If *Type* = KP, inputs *VAL1-VAL4* are defined as follows:

VAL1
First keypoint.

VAL2
Second keypoint.

VAL3
Third keypoint.

VAL4
Arc radius. If *VAL4* = 0 or blank (default), the arc is defined by the three keypoints specified as *VAL1*, *VAL2*, and *VAL3* and arc radius is not used. If *VAL4* is nonzero, *VAL1*, *VAL2*, and *VAL4* are used to calculate the center point, and *VAL3* is used to locate the center point as follows:

VAL4 > 0
Center point and *VAL3* are on the same side of the line between the first two keypoints.

VAL4< 0
Center point and *VAL3* are on opposite sides of the line between the first two keypoints.

If *Type* = LINE, inputs *VAL1-VAL4* are defined as follows:

VAL1
Line number.

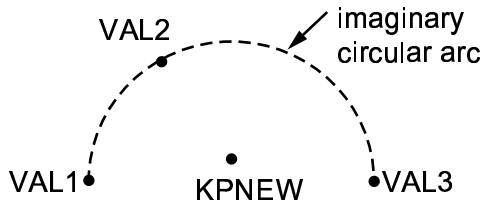
VAL2
Line ratio (0 to 1) indicating the first location (defaults to 0).

VAL3
Line ratio (0 to 1) indicating the second location (defaults to 0.5).

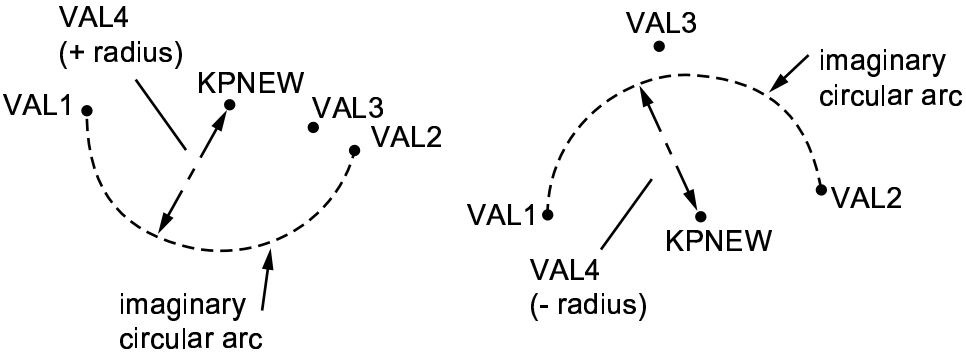
VAL4
Line ratio (0 to 1) indicating the third location (defaults to 1).

Notes

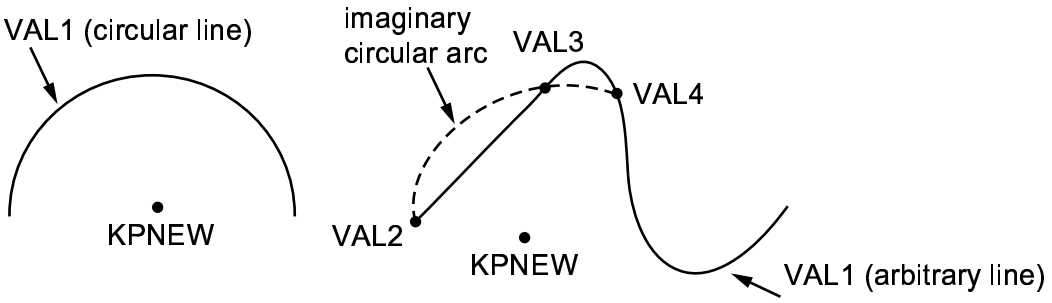
KCENTER should be used in the Cartesian coordinate system (**CSYS**,0) only. This command provides three methods to define a keypoint at the center of three locations. As shown below, the center point can be calculated based on a) three keypoints, b) three keypoints and a radius, or c) three locations on a line. Note that for method c, if a circular line is specified by *VAL1*, *VAL2-VAL4* are not needed.



(a) Three keypoints



(b) Three keypoints and a radius



(c) Locations on line

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>KP at center>3 keypoints
Main Menu>Preprocessor>Modeling>Create>Keypoints>KP at center>3 KPs and radius

Main Menu>Preprocessor>Modeling>Create>Keypoints>KP at center>Location on line

---

KCLEAR, *NP1*, *NP2*, *NINC*

Deletes nodes and point elements associated with selected keypoints.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NP1, *NP2*, *NINC*

Delete mesh for keypoints *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and the mesh for all selected keypoints [**KSEL**] is deleted. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1*.

Notes

Deletes *all* nodes and point elements associated with selected keypoints (regardless of whether the nodes or elements are selected). Nodes associated with non-point elements will not be deleted. Attributes assigned as a result of **KATT** are maintained. In the program's response to the command, if a keypoint is tallied as "cleared," it means either its node or element reference was deleted.

Menu Paths

Main Menu>Preprocessor>Meshing>Clear>Keypoints

KDELE, *NP1*, *NP2*, *NINC*

Deletes unmeshed keypoints.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

NP1, *NP2*, *NINC*

Delete keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and all selected keypoints [**KSEL**] are deleted. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

Notes

Deletes selected keypoints. A keypoint attached to a line cannot be deleted unless the line is first deleted.

Menu Paths

Main Menu>Preprocessor>Modeling>Delete>Keypoints
Main Menu>Preprocessor>Modeling>Topo Repair>Delete>Keypoints

KDIST, *KP1*, *KP2*

Calculates and lists the distance between two keypoints.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

KP1

First keypoint in distance calculation. If *KP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

KP2

Second keypoint in distance calculation.

Notes

KDIST lists the distance between keypoints *KP1* and *KP2*, as well as the current coordinate system offsets from *KP1* to *KP2*, where the X, Y, and Z locations of *KP1* are subtracted from the X, Y, and Z locations of *KP2* (respectively) to determine the offsets. **KDIST** is valid in any coordinate system except toroidal [**CSYS**,3].

KDIST returns a variable, called "*_RETURN*," which contains the distance value. You can use this value for various purposes; for example, to set the default number of line divisions to be generated along region boundary lines [**ESIZE**,*_RETURN*]. In interactive mode, you can access this command by using the Model Query Picker (**Utility Menu> List> Picked Entities**), where you can also access automatic annotation functions, and display the value on your model.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>KP distances

KEEP, *Key*

Stores POST26 definitions and data during active session.

POST26: Display

MP ME ST DY <> PR EM <> FL PP ED

Key

State or value

On or 1

Allows you to exit and reenter **/POST26** without losing your current time history variable information. Keeps a cache of the **/POST26** variable information including the active file name (**FILE**), variable definitions (**NSOL**, **ESOL**, **GAPF**, **RFORCE**, **SOLU**, and **EDREAD**) and stored variable data in memory for the current ANSYS session.

Off or 0

/POST26 variable information is deleted when you exit **/POST26**.

Command Default

ON - Hold time history information in memory. You can, for example, move back and forth between **/POST1** and **/POST26** without redefining and storing the time history variables each time you enter **/POST26**.

Notes

Your variable information is saved in memory only for the current active ANSYS session. It is deleted when you exit ANSYS. This information is also deleted when you issue **/CLEAR**, **RESUME**, **SOLVE**, or **RESET**.

When you reenter **/POST26** all time history variable data is available for use. When you issue **STORE**,*NEW*, variable definitions created by math operations such as **ADD** or **PROD** will not be restored. However, variables defined with **NSOL**, **ESOL**, **GAPF**, **RFORCE**, **SOLU**, and **EDREAD** will be restored. Only the last active results file name is kept in memory (**FILE**).

Commands such as **LAYERP26**, **SHELL**, and **FORCE** that specify the location or a component of data to be stored will retain the setting at the time of exiting **/POST26** .

Menu Paths

Main Menu>TimeHist Postpro>Settings>Data

KESIZE, *NPT*, *SIZE*, *FACT1*, *FACT2*

Specifies the edge lengths of the elements nearest a keypoint.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NPT
Number of the keypoint whose lines will be adjusted. If ALL, use all selected keypoints [**KSEL**]. If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

SIZE
Size of elements along lines nearest keypoint *NPT* (overrides any other specified size). If *SIZE* is zero (or blank), use *FACT1* or *FACT2*.

FACT1
Scale factor applied to a previously defined *SIZE*. Not used if *SIZE* is input.

FACT2
Scale factor applied to the minimum element division at keypoint *NPT* for any attached line. This feature is useful with adaptive mesh refinement. Not used if *SIZE* or *FACT1* is input.

Notes

Affects only the line divisions adjacent to the keypoint on lines not previously assigned divisions by other line commands [**LESIZE**, etc.]. The remaining line divisions are determined from the division nearest the keypoint at the other end of the line (specified by another **KESIZE** command or the **ESIZE** command). Divisions are transferred to the lines during the mesh operation. If smart element sizing is being used [**SMRTSIZE**], **KESIZE** specifications may be overridden (i.e., a smaller element size may be used) to accommodate curvature and small features.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Keypoints>All KPs
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Keypoints>Clr Size
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Keypoints>Picked KPs

---

KEYOPT, *ITYPE*, *KNUM*, *VALUE*

Sets element key options.

PREP7: Element Type

MP ME ST DY <> PR EM <> FL PP ED

ITYPE
Element type number as defined on the **ET** command.

KNUM
Number of the KEYOPT to be defined (KEYOPT(*KNUM*)).

VALUE
Value of this KEYOPT.

Notes

Alternative to inputting KEYOPT values on **ET** command. Must be used if KEYOPT(7) or greater values are to be input. *ITYPE* must first be defined with the **ET** command.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Flow Environment>FLOTRAN Coor Sys
Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Flow Environment>FLOTRAN Coor Sys
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

KEYPTS

Specifies "Keypoints" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

This command cannot be accessed from a menu.

KEYW, *Keyword*, *KEY*

Sets a keyword used by the GUI for context filtering (GUI).

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Keyword
A keyword which, when set to either true or false, changes the behavior of the GUI.

KEY
Keyword switch:

0
Sets the keyword to "false."

1
Sets the keyword to "true."

Notes

Defines a keyword used by the GUI for context filtering. This is a command generated by the GUI and may appear in the log file (**Jobname.LOG**) if the GUI is used. This command is usually *not* typed in directly in an ANSYS session.

This command is valid in any processor.

Menu Paths

Main Menu>Preferences

KFILL, *NP1*, *NP2*, *NFILL*, *NSTRT*, *NINC*, *SPACE*

Generates keypoints between two keypoints.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

NP1, *NP2*
Beginning and ending keypoints for fill-in. *NP1* defaults to next to last keypoint specified, *NP2* defaults to last keypoint specified. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NFILL
Fill *NFILL* keypoints between *NP1* and *NP2* (defaults to |*NP2*-*NP1*|-1). *NFILL* must be positive.

NSTRT
Keypoint number assigned to first filled-in keypoint (defaults to *NP1* + *NINC*).

NINC
Add this increment to each of the remaining filled-in keypoint numbers (may be positive or negative). Defaults to (*NP2*-*NP1*)/(*NFILL* + 1), i.e., linear interpolation.

SPACE
Spacing ratio. Ratio of last division size to first division size. If > 1.0, divisions increase. If < 1.0, divisions decrease. Ratio defaults to 1.0 (uniform spacing).

Notes

Generates keypoints (in the active coordinate system) between two existing keypoints. The two keypoints may have been defined in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Any number of keypoints may be filled in and any keypoint numbering sequence may be assigned.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>Fill between KPs

---

KGEN, *ITIME*, *NP1*, *NP2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

Generates additional keypoints from a pattern of keypoints.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

ITIME

Do this generation operation a total of *ITIME* times, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be more than 1 for generation to occur.

NP1, *NP2*, *NINC*

Generate keypoints from the pattern of keypoints beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and the pattern is all selected keypoints [**KSEL**]. If *NP1* is negative, *NP2* and *NINC* are ignored and the last |*NP1*| keypoints (in sequence from the highest keypoint number) are used as the pattern to be repeated. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

DX, *DY*, *DZ*

Keypoint location increments in the active coordinate system (DR, Dθ, DZ for cylindrical, DR, Dθ, DΦ for spherical).

KINC

Keypoint increment between generated sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM

Specifies if elements and nodes are also to be generated:

0

Generate nodes and point elements associated with the original keypoints, if they exist.

1

Do not generate nodes and elements.

IMOVE

Specifies whether keypoints will be moved or newly defined:

0

Generate additional keypoints as requested with the *ITIME* argument.

1
Move original keypoints to new position retaining the same keypoint numbers (*ITIME*, *KINC*, and *NOELEM* are ignored). Valid only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates additional keypoints (and corresponding mesh) from a given keypoint pattern. The MAT, TYPE, REAL, and ESYS attributes are based upon the keypoints in the pattern and not upon the current settings. Generation is done in the active coordinate system. Keypoints in the pattern may have been defined in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Keypoints

KL, *NL1*, *RATIO*, *NK1*

Generates a keypoint at a specified location on an existing line.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

NL1
Number of the line. If negative, the direction of line (as interpreted for *RATIO*) is reversed. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

RATIO
Ratio of line length to locate keypoint. Must be between 0.0 and 1.0. Defaults to 0.5 (divide the line in half).

NK1
Number to be assigned to keypoint generated at division location (defaults to lowest available keypoint number [**NUMSTR**]).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>On Line
Main Menu>Preprocessor>Modeling>Create>Keypoints>On Line w/Ratio

---

KLIST, *NP1*, *NP2*, *NINC*, *Lab*

Lists the defined keypoints or hard points.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

NP1, *NP2*, *NINC*
List keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL (default), *NP2* and *NINC* are ignored and all selected keypoints [**KSEL**] are listed. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

Lab
Coordinate listing key:

(blank)
List all keypoint information.

COORD
Suppress all but the keypoint coordinates (shown to a higher degree of accuracy than when displayed with all information).

HPT
List only hard point information.

Notes

Lists keypoints in the active display coordinate system [**DSYS**]. An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **KATT** command (and will not be reset to zero if the mesh is cleared); one listed as a negative value indicates that the attribute was assigned using the attribute pointer [TYPE, MAT, REAL, or ESYS] that was active during meshing (and will be reset to zero if the mesh is cleared).

This command is valid in any processor.

Menu Paths

Utility Menu>List>Keypoints>Coordinates +Attributes
Utility Menu>List>Keypoints>Coordinates only
Utility Menu>List>Keypoints>Hard Points

KMESH, *NP1*, *NP2*, *NINC*
Generates nodes and point elements at keypoints.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NP1, *NP2*, *NINC*
Mesh keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and all selected keypoints [**KSEL**] are meshed. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1*.

Notes

Missing nodes required for the generated elements are created and assigned the lowest available numbers.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Keypoints

KMODIF, *NPT*, *X*, *Y*, *Z*

Modifies an existing keypoint.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

NPT

Modify coordinates of this keypoint. If *NPT* = ALL, modify coordinates of all selected keypoints [**KSEL**]. If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

X, *Y*, *Z*

Replace the previous coordinate values assigned to this keypoint with these corresponding coordinate values. Values are interpreted according to the active coordinate system (R, θ, Z for cylindrical, R, θ,Φ for spherical). If *X* = P, graphical picking is used to locate keypoint and *Y* and *Z* are ignored. A blank retains the previous value. You cannot specify *Y* = P.

Notes

Lines, areas, and volumes attached to the modified keypoint (if any) must all be selected and will be redefined using the active coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Caution: Redefined entities may be removed from any defined components and assemblies. Nodes and elements will be automatically cleared from any redefined keypoints, lines, areas, or volumes.

The **KMODIF** command moves keypoints for geometry modification without validating underlying entities. To merge keypoints and update higher order entities, issue the **NUMMRG** command instead.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Keypoints>Set of KPs
Main Menu>Preprocessor>Modeling>Move / Modify>Keypoints>Single KP

KMOVE, *NPT*, *KC1*, *X1*, *Y1*, *Z1*, *KC2*, *X2*, *Y2*, *Z2*

Calculates and moves a keypoint to an intersection.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

NPT

Move this keypoint. If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

KC1

First coordinate system number. Defaults to 0 (global Cartesian).

X1, *Y1*, *Z1*

Input one or two values defining the location of the keypoint in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Fields are R1, θ1, Z1 for cylindrical, or R1, θ1, ϕ1 for spherical.

KC2

Second coordinate system number.

X2, *Y2*, *Z2*

Input two or one value(s) defining the location of the keypoint in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Arguments are R2, θ2, Z2 for cylindrical, or R2, θ2, ϕ2 for spherical.

Notes

Calculates and moves a keypoint to an intersection location. The keypoint must have been previously defined (at an approximate location) or left undefined (in which case it is internally defined at the **SOURCE** location). The actual location is calculated from the intersection of three surfaces (implied from three coordinate constants in two different coordinate systems). Note that solid modeling in a toroidal coordinate system is not recommended. See the **MOVE** command for surface and intersection details. The three (of six) constants easiest to define should be used. The program will calculate the remaining three coordinate constants. All arguments, except *KC1*, must be input. Use the repeat command [***REPEAT**] after the **KMOVE** command to move a series of keypoints, if desired.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Keypoints>To Intersect

---

KNODE, *NPT*, *NODE*

Defines a keypoint at an existing node location.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

NPT

Arbitrary reference number for keypoint. If zero, the lowest available number is assigned [**NUMSTR**].

NODE

Node number defining global X, Y, Z keypoint location. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>On Node

KPLOT, *NP1*, *NP2*, *NINC*, *Lab*

Displays the selected keypoints.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

NP1, *NP2*, *NINC*

Display keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL (default), *NP2* and *NINC* are ignored and all selected keypoints [**KSEL**] are displayed.

Lab

Determines what keypoints are plotted (one of the following):

(blank)
Plots all keypoints.

HPT
Plots only those keypoints that are hard points.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Keypoints>Hardpoints
Utility Menu>Plot>Keypoints>Keypoints
Utility Menu>Plot>Specified Entities>Keypoints

KPSCALE, *NP1*, *NP2*, *NINC*, *RX*, *RY*, *RZ*, *KINC*, *NOELEM*, *IMOVE*

Generates a scaled set of (meshed) keypoints from a pattern of keypoints.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

NP1, *NP2*, *NINC*
Set of keypoints (*NP1* to *NP2* in steps of *NINC*) that defines the pattern to be scaled. *NP2* defaults to *NP1*, *NINC* defaults to 1. If *NP1* = ALL, *NP2* and *NINC* are ignored and the pattern is defined by all selected keypoints. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

RX, *RY*, *RZ*
Scale factors to be applied to the X, Y, Z keypoint coordinates in the active coordinate system (RR, Rθ, RZ for cylindrical; RR, Rθ, RΦ for spherical). The Rθ and RΦ scale factors are interpreted as angular *offsets*. For example, if CSYS = 1, an *RX*, *RY*, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints.) Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

KINC
Increment to be applied to the keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned [**NUMSTR**].

NOELEM
Specifies whether nodes and elements are also to be generated:

0
Nodes and point elements associated with the original keypoints will be generated (scaled) if they exist.

1
Nodes and point elements will *not* be generated.

IMOVE
Specifies whether keypoints will be moved or newly defined:

0
Additional keypoints will be generated.

1
Original keypoints will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a scaled set of keypoints (and corresponding mesh) from a pattern of keypoints. The MAT, TYPE, REAL, and ESYS attributes are based on the keypoints in the pattern and not the current settings. Scaling is done in the active coordinate system. Keypoints in the pattern could have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Scale>Keypoints

KREFINE, *NP1*, *NP2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

Refines the mesh around specified keypoints.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NP1, *NP2*, *NINC*
Keypoints (*NP1* to *NP2* in increments of *NINC*) around which the mesh is to be refined. *NP2* defaults to *NP1*, and *NINC* defaults to 1. If *NP1* = ALL, *NP2* and *NINC* are ignored and all selected keypoints are used for refinement. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

LEVEL
Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

DEPTH
Depth of mesh refinement in terms of the number of elements outward from the indicated keypoints (defaults to 1).

POST
Type of postprocessing to be done after element splitting, in order to improve element quality:

OFF
No postprocessing will be done.

SMOOTH
Smoothing will be done. Node locations may change.

CLEAN
Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

RETAIN
Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The ANSYS program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

ON
The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

OFF
The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

Notes

KREFINE performs local mesh refinement around the specified keypoints. By default, the indicated elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

KREFINE refines all area elements and tetrahedral volume elements that are adjacent to the specified keypoints. Any volume elements that are adjacent to the specified keypoints, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes [**IC**], coupled nodes [**CP** family of commands], constraint equations [**CE** family of commands], or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. See Revising Your Model in the *ANSYS Modeling and Meshing Guide* for additional restrictions on mesh refinement.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Keypoints

---

KSCALE, *KINC*, *NP1*, *NP2*, *NINC*, *RX*, *RY*, *RZ*

Generates a scaled pattern of keypoints from a given keypoint pattern.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

KINC
Do this scaling operation one time, incrementing all keypoints in the given pattern by *KINC*. If *KINC* = 0, keypoints will be redefined at the scaled locations.

NP1, *NP2*, *NINC*
Scale keypoints from pattern beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and pattern is all selected keypoints [**KSEL**]. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

RX, *RY*, *RZ*
Scale factor ratios. Scaling is relative to the origin of the active coordinate system (RR, Rθ, RZ for cylindrical, RR, Rθ, RΦ for spherical). If > 1.0, pattern is enlarged. If < 1.0, pattern is reduced. Ratios each default to 1.0.

Notes

Generates a scaled pattern of keypoints from a given keypoint pattern. Scaling is done in the active coordinate system (see analogous node scaling [**NSCALE**]). Solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

This command cannot be accessed from a menu.

KSCON, *NPT*, *DELR*, *KCTIP*, *NTHET*, *RRAT*

Specifies a keypoint about which an area mesh will be skewed.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NPT
Keypoint number at concentration. If *NPT* = ALL, use all selected keypoints. If remaining fields are blank, remove concentration from this keypoint (if unmeshed). If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

DELR
Radius of first row of elements about keypoint.

KCTIP
Crack tip singularity key:

0
Do not skew midside nodes, if any, within the element.

1
Skew midside nodes of the first row of elements to the 1/4 point for crack tip singularity.

NTHET
Number of elements in circumferential direction (defaults to approximately one per 45° (or one per 30°, if *KCTIP* = 1)).

RRAT
Ratio of 2nd row element size to *DELR* (defaults to 0.75, or 0.5 if *KCTIP* = 1).

Notes

Defines a concentration keypoint about which an area mesh will be skewed. Useful for modeling stress concentrations and crack tips. During meshing, elements are initially generated circumferentially about, and radially away, from the keypoint. Lines attached to the keypoint are given appropriate divisions and spacing ratios. Only one concentration keypoint per unmeshed area is allowed. Use **KSCON**,STAT to list current status of concentration keypoints. The **KSCON** command does not support 3-D modeling.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>Concentrat KPs>Create
Main Menu>Preprocessor>Meshing>Size Cntrls>Concentrat KPs>List

---

KSEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS*

Selects a subset of keypoints or hard points.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type

Label identifying the type of select:

S
: Select a new set (default).

R
: Reselect a set from the current set.

A
: Additionally select a set and extend the current set.

U
: Unselect a set from the current set.

ALL
: Restore the full set.

NONE
: Unselect the full set.

INVE
: Invert the current set (selected becomes unselected and vice versa).

STAT
: Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Item

Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to KP.

Comp

Component of the item (if required). Valid component labels are shown in the table below.

VMIN

Minimum value of item range. Ranges are keypoint numbers, coordinate values, attribute numbers, etc., as appropriate for the item. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, or ESYS and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **KLIST** command for a discussion of signed attributes.

VMAX

Maximum value of item range. *VMAX* defaults to *VMIN*. If *VMIN* = *VMAX*, a tolerance of ±0.005x*VMIN* is used, or ±1.0E-6 if *VMIN* = 0.0. If *VMAX* ≠ *VMIN*, a tolerance of ±1.0E-8x(*VMAX*-*VMIN*) is used.

VINC

Value increment within range. Used only with integer ranges (such as for keypoint numbers). Defaults to 1. *VINC* cannot be negative.

KABS
Absolute value key:

0
Check sign of value during selection.

1
Use absolute value during selection (sign ignored).

Command Default

All keypoints are selected.

Notes

Selects a subset of keypoints or hard points. For example, to select a new set of keypoints based on keypoint numbers 1 through 7, use **KSEL**,S,KP,,1,7. The selected subset is used when the ALL label is entered (or implied) on other commands, such as **KLIST**,ALL. Only data identified by keypoint number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

This command is valid in any processor.

KSEL - Valid Item and Component Labels

Valid Item and Component Labels KSEL, *Type,Item,Comp,VMIN,VMAX,VINC,KABS*

Item	Comp	Description
KP		Keypoint number.
EXT		Keypoint numbers on exterior of selected lines (ignore remaining fields).
HPT		Hard point number.
LOC	X,Y,Z	X, Y, or Z location in the active coordinate system.
MAT		Material number associated with the keypoint.
TYPE		Element type number associated with the keypoint.
REAL		Real constant set number associated with the keypoint.
ESYS		Element coordinate system associated with the keypoint.

Menu Paths

Utility Menu>Select>Entities

KSLL, *Type*

Selects those keypoints contained in the selected lines.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of keypoint select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

KSLN, *Type*

Selects those keypoints associated with the selected nodes.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of keypoint select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

Notes

Valid only if the nodes were generated by a *meshing* operation [**KMESH**, **LMESH**, **AMESH**, **VMESH**] on a solid model that contains the associated keypoints.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

KSUM

Calculates and prints geometry statistics of the selected keypoints.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

Notes

Calculates and prints geometry statistics (centroid location, moments of inertia, etc.) associated with the selected keypoints. Geometry items are reported in the global Cartesian coordinate system. A unit density is assumed, irrespective of any material associations [**KATT**, **MAT**]. Items calculated by **KSUM** and later retrieved by a ***GET** or ***VGET** command are valid only if the model is not modified after the **KSUM** command is issued.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Keypoints

---

KSYMM, *Ncomp*, *NP1*, *NP2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Generates a reflected set of keypoints.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

Ncomp

Symmetry key:

X

X (or R) symmetry (default).

Y

Y (or θ) symmetry.

Z

Z (or Φ) symmetry.

NP1, *NP2*, *NINC*

Reflect keypoints from pattern beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and pattern is all selected keypoints [**KSEL**]. If *Ncomp* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

KINC

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Generate nodes and point elements associated with the original keypoints, if they exist.

1

Do not generate nodes and elements.

IMOVE
Specifies whether keypoints will be moved or newly defined:

0
Generate additional keypoints.

1
Move original keypoints to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a reflected set of keypoints (and corresponding mesh) from a given keypoint pattern by a symmetry reflection (see analogous node symmetry command, **NSYM**). The MAT, TYPE, REAL, and ESYS attributes are based upon the keypoints in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. Keypoints in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Keypoints

KTRAN, *KCNTO*, *NP1*, *NP2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Transfers a pattern of keypoints to another coordinate system.

PREP7: Keypoints

MP ME ST DY <> PR EM <> FL PP ED

KCNTO
Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system.

NP1, *NP2*, *NINC*
Transfer keypoints from pattern beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and pattern is all selected keypoints [**KSEL**]. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

KINC
Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM
Specifies whether nodes and elements are also to be generated:

0
Generate nodes and point elements associated with the original keypoints, if they exist.

1
Do not generate nodes and elements.

IMOVE
Specifies whether keypoints will be moved or newly defined:

0
Generate additional keypoints.

1
Move original keypoints to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Transfers a pattern of keypoints (and corresponding mesh) from one coordinate system to another (see analogous node transfer command, **TRANSFER**). The MAT, TYPE, REAL, and ESYS attributes are based upon the keypoints in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. Coordinate values are interpreted in the active coordinate system and are transferred directly. Solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Keypoints

---

KUSE, *KEY*

Specifies whether or not to reuse the triangularized matrix.

SOLUTION: Load Step Options

MP ME ST <> <> PR EM <> <> PP ED

KEY
Reuse key:

0
Program decides whether or not to reuse the previous triangularized stiffness matrix.

1
Force the previous triangularized stiffness matrix to be reused. Used mainly in a restart. Forcing reuse of the matrix is a nonstandard use of the program, and should be done with caution. For instance, using this option and changing the number of elements, or the number or type of degrees of freedom, may cause an abort.

1
All element matrices are reformed and are used to reform a new triangularized stiffness matrix.

Command Default

Program makes decision.

Notes

Overrides the program logic to determine whether or not to reuse the previous triangularized stiffness matrix for each substep of this load step. Applies only to static or full transient analyses and to full harmonic analyses if the frequency is not changed for continuing loadsteps. For full harmonic analyses, only *KEY* = 1 or *KEY* = 0 is valid.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Reuse Tri Matrix
Main Menu>Solution>Load Step Opts>Other>Reuse Tri Matrix

KWPAVE, *P1*, *P2*, *P3*, *P4*, *P5*, *P6*, *P7*, *P8*, *P9*

Moves the working plane origin to the average location of keypoints.

DATABASE: Working Plane

MP ME ST DY <> PR EM <> FL PP ED

P1, *P2*, *P3*, *P4*, *P5*, *P6*, *P7*, *P8*, *P9*
Keypoints used in calculation of the average. At least one must be defined. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Moves the origin of the working plane to the average of the specified keypoints. Averaging is based on the active coordinate system.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Offset WP to>Keypoints

KWPLAN, *WN*, *KORIG*, *KXAX*, *KPLAN*

Defines the working plane using three keypoints.

DATABASE: Working Plane

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified. If fewer than three points are used, the viewing direction of window *WN* will be used instead to define the normal to the working plane.

KORIG
Keypoint number defining the origin of the working plane coordinate system. If *KORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

KXAX
Keypoint number defining the x-axis orientation (defaults to the x-axis being parallel to the global X-axis; or if the normal to the working plane is parallel to the global X-axis, then defaults to being parallel to the global Y-axis).

KPLAN
Keypoint number defining the working plane (the normal defaults to the present display view [**/VIEW**] of window *WN*).

Notes

Defines a working plane to assist in picking operations using three keypoints as an alternate to the **WPLANE** command. The three keypoints also define the working plane coordinate system. A minimum of one keypoint (at the working plane origin) is required. Immediate mode may also be active. See **WPSTYL** command to set the style of working plane display.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Align WP with>Keypoints

L Commands

L, *P1*, *P2*, *NDIV*, *SPACE*, *XV1*, *YV1*, *ZV1*, *XV2*, *YV2*, *ZV2*
Defines a line between two keypoints.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

P1
Keypoint at the beginning of line. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P2
Keypoint at the end of line.

NDIV
Number of element divisions within this line. Normally this field is not used; specifying divisions with **LESIZE**, etc. is recommended.

SPACE
Spacing ratio. Normally this field is not used, as specifying spacing ratios with the **LESIZE** command is recommended. If positive, *SPACE* is the nominal ratio of the last division size (at *P2*) to the first division size (at *P1*). If the ratio is greater than 1, the division sizes increase from *P1* to *P2*, and if less than 1, they decrease. If *SPACE* is negative, then |*SPACE*| is the nominal ratio of the center division size to those at the ends.

The following fields are used only if specified end slopes on the line are desired, otherwise zero curvature end slopes will be automatically calculated to produce a line which is "straight" in the active coordinate system. To specify end slopes, use the following fields to define a "slope vector" (one for each end of the line, if desired) that has its tail at the origin and its head at the point *XV*,*YV*,*ZV* in the active coordinate system [**CSYS**]. The corresponding end slope of the line will then be parallel to this "slope vector."

XV1, *YV1*, *ZV1*
Location (in the active coordinate system) of the head of the "slope vector" corresponding to the slope at the *P1* end of the line. The tail of the vector is at the origin of the coordinate system.

XV2, *YV2*, *ZV2*
Location of the head of the "slope vector" corresponding to the slope at the *P2* end of the line.

Notes

Defines a line between two keypoints from *P1* to *P2*. The line shape may be generated as "straight" (in the active coordinate system) or curved. The line shape is invariant with coordinate system after it is generated. Note that solid modeling in a toroidal coordinate system is not recommended. A curved line is limited to 180°. Lines may be redefined only if not yet attached to an area.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>In Active Coord

L2ANG, *NL1*, *NL2*, *ANG1*, *ANG2*, *PHIT1*, *PHIT2*

Generates a line at an angle with two existing lines.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1
Number of the first line to be hit (touched by the end of the new line). If negative, assume *P1* (see below) is the second keypoint of the line instead of the first. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NL2
Number of the second line to be hit. If negative, assume *P3* is the second keypoint of the line instead of the first.

ANG1
Angle of intersection (usually zero or 180) of generated line with tangent to first line.

ANG2
Angle of intersection (usually zero or 180) of generated line with tangent to second line.

PHIT1
Number to be assigned to keypoint generated at hit location on first line (defaults to lowest available keypoint number [**NUMSTR**]).

PHIT2
Number to be assigned to keypoint generated at hit location on second line (defaults to lowest available keypoint number [**NUMSTR**]).

Notes

Generates a straight line (*PHIT1-PHIT2*) at an angle (*ANG1*) with an existing line *NL1* (*P1-P2*) and which is also at an angle (*ANG2*) with another existing line *NL2* (*P3-P4*). If the angles are zero the generated line is tangent to the two lines. The *PHIT1* and *PHIT2* locations on the lines are automatically calculated. Line *P1-P2* becomes *P1-PHIT1*, *P3-P4* becomes *P3-PHIT2*, and new lines *PHIT1-P2*, *PHIT2-P4*, and *PHIT1-PHIT2* are generated. Line divisions are set to zero (use **LESIZE**, etc. to modify).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Angle to 2 Lines
Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Norm to 2 Lines

---

L2TAN, *NL1*, *NL2*

Generates a line tangent to two lines.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1
Number of the first line generated line is tangent to. If negative, assume *P1* (see below) is the second keypoint of the line instead of the first. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NL2
Number of the second line generated line is tangent to. If negative, assume *P3* is the second keypoint of the line instead of the first.

Notes

Generates a line (*P2-P3*) tangent at point *P2* to line *NL1* (*P1-P2*) and tangent at point *P3* to line *NL2* (*P3-P4*).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Tan to 2 Lines

---

LANG, *NL1*, *P3*, *ANG*, *PHIT*, *LOCAT*

Generates a straight line at an angle with a line.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1
Number of the line to be hit (touched by the end of the new line). If negative, assume *P1* (see below) is the second keypoint of the line instead of the first. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P3
Keypoint at which generated line must end.

ANG
Angle of intersection of generated line *PHIT-P3* with tangent to line *P1-P2* at *PHIT*. If 0 (default), the generated line is tangent to *NL1* toward end P1; if 90, the generated line is perpendicular to *NL1*. If 180, the generated line is tangent to NL1 toward end P2. *ANG* can be any value, but is adjusted to the corresponding acute angle with respect to *LOCAT*. See Notes for a discussion of accuracy.

PHIT
Number to be assigned to keypoint generated at hit location (defaults to lowest available keypoint number [**NUMSTR**]).

LOCAT
Approximate location of *PHIT* in terms of the ratio of the distance along the line (*NL1*) to the length of the line. *LOCAT* can range from 0 to 1. If *LOCAT* is blank, the point will be located with less speed and accuracy, and an arbitrary location may result.

Notes

Generates a straight line (*PHIT-P3*) at an angle (*ANG*) with a line *NL1* (*P1-P2*). The location of *PHIT* on the line is automatically calculated. Line *P1-P2* becomes *P1-PHIT* and new lines *PHIT-P2* and *PHIT-P3* are generated. Line divisions are set to zero (use **LESIZE**, etc. to modify).

PHIT is positioned closest to *LOCAT* for the given angle, *ANG*. To ensure better performance, it is recommended that *LOCAT* be input, even if it is 0.

The program uses an iterative procedure to position *PHIT*. The procedure is not exact, with the result that the actual value of *ANG* will sometimes differ slightly from the specified value.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>At angle to line
Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Normal to Line

---

LARC, *P1*, *P2*, *PC*, *RAD*

Defines a circular arc.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

P1
Keypoint at one end of circular arc line. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P2
Keypoint at other end of circular arc line.

PC
Keypoint defining plane of arc and center of curvature side (with positive radius). Must not lie along the straight line from *P1* to *P2*. *PC* need not be at the center of curvature.

RAD
Radius of curvature of the arc. If negative, assume center of curvature side is opposite to that defined by *PC*. If *RAD* is blank, *RAD* will be calculated from a curve fit through *P1*, *PC*, and *P2*.

Notes

Defines a circular arc line from *P1* to *P2*. The line shape is generated as circular, regardless of the active coordinate system. The line shape is invariant with coordinate system after it is generated.

When dealing with a large radius arc (1e3), or if the location of the arc you create is far away from the origin of your coordinate system, anomalies may occur. You can prevent this by creating the arc at a smaller scale, and then scaling the model back to full size (**LSSCALE**).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>By End KPs & Rad
Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>Through 3 KPs

---

/LARC, *XCENTR*, *YCENTR*, *XLRAD*, *ANGLE1*, *ANGLE2*

Creates annotation arcs (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

XCENTR
Arc X center location (-1.0 < X < 1.0).

YCENTR
Arc Y center location (-1.0 < Y < 1.0).

XLRAD
Arc radius length.

ANGLE1
Starting angle of arc.

ANGLE2
Ending angle of arc. The arc is drawn counterclockwise from the starting angle, *ANGLE1*, to the ending angle, *ANGLE2*.

Notes

Defines annotation arcs to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All arcs are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** command to set the attributes of the arc.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

LAREA, *P1*, *P2*, *NAREA*

Generates the shortest line between two keypoints on an area.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

P1
First keypoint of line to be generated. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P2
Second keypoint of line to be generated.

NAREA
Area containing *P1* and *P2*, or area to which generated line is to be parallel.

Notes

Generates the shortest line between two keypoints, *P1* and *P2*, both of which lie on an area. The generated line will also lie on the area. *P1* and *P2* may also be equidistant (in global Cartesian space) from the area (and on the same side of the area), in which case a line parallel to the area is generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Overlaid on Area

---

LARGE, *IR*, *IA*, *IB*, *IC*, *Name*, --, --, *FACTA*, *FACTB*, *FACTC*

Finds the largest (the envelope) of three variables.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA, *IB*, *IC*

Reference numbers of the three variables to be operated on. If only two, leave *IC* blank. If only one, leave *IB* blank also.

Name

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --

Unused fields.

FACTA, *FACTB*, *FACTC*

Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

Notes

Creates a new variable by finding the largest of up to three variables according to the operation:

IR = Largest of (*FACTA* x *IA*, *FACTB* x *IB*, *FACTC* x *IC*)

The comparison is done at each time location, so that the new variable is the "envelope" of the three existing variables.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Find Maximum

LATT, *MAT*, *REAL*, *TYPE*, --, *KB*, *KE*, *SECNUM*

Associates element attributes with the selected, unmeshed lines.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

MAT, *REAL*, *TYPE*

Material number, real constant set number, and type number to be associated with selected, unmeshed lines.

--

Unused field.

KB, *KE*

Beginning and ending orientation keypoints to be associated with selected, unmeshed lines. ANSYS uses the location of these keypoints to determine how to orient beam cross sections during beam meshing. Beam elements may be created along a line with a constant orientation by specifying only one orientation keypoint

(*KB*), or a pre-twisted beam may be created by selecting different orientation keypoints at each end of the line (*KB* and *KE*). (For a line bounded by two keypoints (*KP1* and *KP2*), the orientation vector at the beginning of the line extends from *KP1* to *KB*, and the orientation vector at the end of the line extends from *KP2* to *KE*. The orientation vectors are used to compute the orientation nodes of the elements.)

SECNUM

Section identifier to be associated with selected, unmeshed lines. For details, see the description of the **SECTYPE** and **SECNUM** commands.

Notes

The element attributes specified by the **LATT** command will be used when the lines are meshed.

Lines subsequently generated from the lines will also have the attributes specified by *MAT*, *REAL*, *TYPE*, and *SECNUM*. If a line does not have these attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT**, **REAL**, **TYPE**, and **SECNUM** command settings.

In contrast, the values specified by *KB* and *KE* apply only to the selected lines; that is, lines subsequently generated from these lines will not share these attributes. Similarly, if a line does not have *KB* and *KE* attributes associated with it via the **LATT** command at the time it is meshed, ANSYS cannot obtain the attributes from elsewhere. See the discussion on beam meshing in Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide* for more information.

Reissue the **LATT** command (before lines are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association. If any of the arguments *MAT*, *REAL*, *TYPE*, or *ESYS* are defined as -1, then that value will be left unchanged in the selected set.

In some cases, ANSYS can proceed with a line meshing operation even when no logical element type has been assigned via **LATT**,,,*TYPE* or **TYPE**. See Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide* for more information about setting element attributes.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>All Lines
Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked Lines

LAYER, *NUM*

Specifies the element layer for which data are to be processed.

POST1: Controls

MP ME ST DY <> PR <> <> <> PP <>

NUM

Layer number for layered element types (SOLID46, SHELL91, SHELL99, SHELL131, SHELL132, SHELL163, SHELL181, SOLID191, SHELL208, or SHELL209). See Notes below for SHELL163 usage.

If SOLID46, SHELL91, SHELL99, or SOLID191 was used and the **TB**,FAIL command was used to specify failure criteria, *NUM* = FCMAX can be used to process the maximum failure criteria in the element, regardless of layer number. This *NUM* = FCMAX option allows the user to process the maximum failure criteria (and the stresses and strains that created it) as constant quantities over the entire element; thus for this case, the **SHELL** command has no function.

Command Default

The entire element is considered to be the default "Layer". Accordingly, the results data are from the bottom of the bottom layer and the top of the top layer.

Notes

Specifies the element layer for which results data are to be listed, plotted, or otherwise processed. Applies to stress and strain data for layered elements SOLID46, SHELL91, SHELL99, SHELL163, SHELL181, SOLID191, SHELL208, and SHELL209; heat flux and heat gradient for SHELL131 and SHELL132; body load temperatures for SHELL91 and SOLID191. The **SHELL** command may then be used (with shell elements) to specify a location (TOP, MID, BOT) within the layer for output. (The **SHELL** command does not apply to thermal shell elements SHELL131 and SHELL132.) Transverse shear stresses for MID are linearly averaged from TOP and BOT, and do not reflect a parabolic distribution. Setting KEYOPT(8) = 2 for SHELL181, SHELL208, and SHELL209, writes the mid-surface values directly to the results file and yields more accurate values than linear averaging. Also note that this command cannot be used for energy output, as energy is a per-element quantity. When using the **LAYER** command with SOLID46, SHELL91, SHELL99, SHELL181, SOLID191, SHELL208, and SHELL209, KEYOPT(8) must be set to 1 (or 2 for SHELL181, SHELL208, or SHELL209) in order to store results for all layers.

Use **RSYS**,SOLU to display results in the layer coordinate system for a particular layer.

For the ANSYS LS-DYNA product, this command works differently than described above. For SHELL163, you must first use **EDINT** during the solution phase to define the integration points for which you want output data. Be aware that the output location for SHELL163 data is always at the integration point, so "top" and "bottom" refer to the top or bottom integration point, not necessarily the top or bottom surface. For more information, see the *ANSYS LS-DYNA User's Guide*.

Menu Paths

Main Menu>General Postproc>Options for Outp
Utility Menu>List>Results>Options

LAYERP26, *NUM*

Specifies the element layer for which data are to be stored.

POST26: Controls

MP ME ST DY <> PR <> <> <> PP <>

NUM

Layer number for layered element types (SOLID46, SHELL91, SHELL99, BEAM161, SHELL163, SHELL181, SOLID191, SHELL208, or SHELL209). See Notes below for BEAM161 and SHELL163 usage.

If SOLID46, SHELL91, SHELL99, or SOLID191 was used and the **TB**,FAIL command was used to specify failure criteria, *NUM* = FCMAX can be used to process the maximum failure criteria in the element, regardless of layer number. This *NUM* = FCMAX option allows the user to process the maximum failure criteria (and the stresses and strains that created it) as constant quantities over the entire element; thus for this case, the SHELL command has no function.

This command is also applicable to SHELL163. See notes below for details.

Command Default

Results data are from the "first" layer. If KEYOPT(8) = 0, "first" layer actually means the bottom of the bottom layer and the top of the top layer.

Notes

Defines the element layer for which results data are to be stored for postprocessing. Applies to stress and strain data for layered elements SOLID46, SHELL91, SHELL99, BEAM161, SHELL163, SHELL181, SOLID191, SHELL208, and SHELL209. Also applies to body load temperatures for SHELL99. The **SHELL** command may then be used (for shell elements) to specify a location (TOP, MID, BOT) within the layer for selection on the **ESOL** command. Transverse shear stresses for MID are linearly averaged from TOP and BOT, and do not reflect a parabolic distribution. Setting KEYOPT(8) = 2 for SHELL181, SHELL208, and SHELL209, writes the mid-surface values directly to the results file and yields more accurate values than linear averaging. Also note that this command cannot be used for energy output, as energy is a per-element quantity. When using the **LAYERP26** command with SOLID46, SHELL91, SHELL99, SHELL181, SOLID191, SHELL208, or SHELL209, KEYOPT(8) must be set to 1 (or 2 for SHELL181, SHELL208, or SHELL209) in order to store results for all layers.

For the ANSYS LS-DYNA product, this command works differently than described above. For SHELL163 and BEAM161, you must first use **EDINT** during the solution phase to define the integration points for which you want output data. Be aware that the output location for SHELL163 data is always at the integration point, so "top" and "bottom" refer to the top or bottom integration point, not necessarily the top or bottom surface. For more information, see the *ANSYS LS-DYNA User's Guide*.

In POST26, the **ESOL** data stored is based on the active **LAYERP26** specification at the time the data is stored. To store data at various specifications (for example, layers 2 and 5), issue a **STORE** command before each new specification.

Menu Paths

Main Menu>TimeHist Postpro>Define Variables
Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

LAYLIST, *IEL*, *LAYR1*, *LAYR2*, *Mplab1*, *Mplab2*

Lists real constants material properties for layered elements.

PREP7: Elements

MP ME ST DY <> PR <> <> <> PP <>

IEL
Element number to be listed. If ALL, list all selected elements [**ESEL**] of the appropriate type. If blank and the current element type is a layered element type, list data from the current real constant table in the layered format.

LAYR1, *LAYR2*
Range of layer numbers to be listed. If *LAYR1* is greater than *LAYR2*, a reverse order list is produced. *LAYR1* defaults to 1. *LAYR2* defaults to *LAYR1* if *LAYR1* is input or to the number of layers if *LAYR1* is not input.

Mplab1, *Mplab2*
Material property labels (e.g., EX) to be listed along with the layer real constants.

Notes

Lists real constants and any two material properties for layered shell and solid elements. Only elements of type SOLID46, SHELL91, SHELL99, and SOLID191 are listed.

If matrix input is selected (KEYOPT(2) = 2 or 3), *LAYR1*, *LAYR2*, *Mplab1*, and *Mplab2* are not used.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Elements>Layered Elements
Utility Menu>List>Properties>Layer Data

LAYPLOT, *IEL*, *LAYR1*, *LAYR2*

Displays the layer stacking sequence for layered elements.

PREP7: Elements

MP ME ST DY <> PR <> <> <> PP <>

IEL
Element number for the display. If blank and the current element type is a layered element type, display data from the current real constant table.

LAYR1, *LAYR2*
Range of layer numbers to be displayed. If *LAYR1* is greater than *LAYR2*, a reversed order display is produced. Up to 20 layers may be displayed at a time. *LAYR1* defaults to 1. *LAYR2* defaults to *LAYR1* if *LAYR1* is input or to the number of layers (or to 19+*LAYR1*, if smaller) if *LAYR1* is not input.

Notes

Displays the layer stacking sequence as defined in the real constant table for layered shell and solid elements in a form where the layers are visible (like a sheared deck of cards). The element x-axis is shown as 0.0 degrees. Layers are cross-hatched and color coded for clarity. The hatch lines indicate the layer angle (real constant THETA) and the color coding is for material identification (real constant MAT). Only valid for elements of type SOLID46, SHELL91, SHELL99, and SOLID191. The actual orientation of a specific layer in three dimensional space can be seen using **/PSYMB**, LAYR. Layer thickness can be displayed using the **/ESHAPE** and **EPLOT** commands.

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Layered Elements

LCABS, *LCNO*, *KABS*

Specifies absolute values for load case operations.

POST1: Load Case Calculations

MP ME ST DY <> PR EM <> <> PP ED

LCNO
Load case pointer number. If ALL, apply to all selected load cases [**LCSEL**].

KABS
Absolute value key:

0
Use algebraic values of load case *LCNO* in operations.

1
Use absolute values of load case *LCNO* in operations.

Command Default

Use algebraic values.

Notes

Causes absolute values to be used in the load case operations [**LCASE** or **LCOPER**]. Absolute values are taken prior to assigning a load case factor [**LCFACT**] and are applied only to defined load cases [**LCDEF**].

Menu Paths

Main Menu>General Postproc>Load Case>Calc Options>Absolut Value

---

LCASE, *LCNO*

Reads a load case into the database.

POST1: Load Case Calculations

MP ME ST DY <> PR EM <> <> PP ED

LCNO
Load case pointer number [**LCDEF**,STAT]. Defaults to 1.

Command Default

Load case 1.

Notes

Reads a load case into the database. Load cases are created as described on the **LCDEF** or **LCWRITE** commands. The results portion of the database and the applied forces and displacements are cleared before reading the data in. Absolute values [**LCABS**] and scale factors [**LCFACT**] can be applied during the read operation.

Menu Paths

Main Menu>General Postproc>Load Case>Read Load Case

LCCALC

Specifies "Load case settings" as the subsequent status topic.

POST1: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Load Case Calcs

LCCAT, *NL1*, *NL2*

Concatenates multiple lines into one line for mapped meshing.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*

Lines to be concatenated. If *NL1* = ALL, *NL2* is ignored and all selected lines [**LSEL**] are concatenated. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* is ignored).

Notes

Concatenates multiple, adjacent lines (the input lines) into one line (the output line) in preparation for mapped meshing. An area that contains too many lines for mapped meshing can still be mapped meshed if some of the lines in that area are first concatenated (see Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide* for details on mapped meshing restrictions).

LCCAT is meant to be used solely for meshing and cannot be used for any other purposes. Specifically, (a) the output line and any areas that have the output line on their line list [**ALIST**] cannot be used as input to any other solid modeling operation (not even another **LCCAT** command); and (b) the output line cannot accept solid model boundary conditions [**DL**, **SFL**].

The output line will take on the element divisions of the input lines and will not accept element divisions that are directly assigned [**LESIZE**]. The output line from the **LCCAT** operation will be coincident with the input lines and the input lines will be retained. Consider the **LCOMB** command instead of **LCCAT** if you wish to delete the input lines and if the lines to be combined have similar slopes at the common keypoint(s). When an **LCCAT**

command is issued, area line lists [**ALIST**] that contain *all* of the input lines will be updated so that the area line lists refer to the output line instead of the input lines. Deletion of the output line [**LDELE**] effectively reverses the **LCCAT** operation and restores area line lists to their original condition.

You can use the **LSEL** command to select lines that were created by concatenation, and then follow it with an **LDELE**,ALL command to delete them. Also see Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide* for a discussion on how to easily select and delete concatenated lines in one step.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>Concatenate>Lines
Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>Concatenate>Lines

---

LCDEF, *LCNO*, *LSTEP*, *SBSTEP*, *KIMG*

Creates a load case from a set of results on a results file.

POST1: Load Case Calculations

MP ME ST DY <> PR EM <> <> PP ED

LCNO
Arbitrary pointer number (1-99) to be assigned to the load case specified by *LSTEP*, *SBSTEP* and by the **FILE** command. Defaults to 1 + previous value.

LSTEP
Load step number to be defined as the load case. Defaults to one.

SBSTEP
Substep number. Defaults to the last substep of the load step.

KIMG
Used only with results from complex analyses:

0
Use real part of complex solution

1
Use imaginary part.

Notes

Creates a load case by establishing a pointer to a set of results on a results file (written during the ANSYS solution phase). This pointer (*LCNO*) can then be used on the **LCASE** or **LCOPER** commands to read the load case data into the database.

Issue **LCDEF**,ERASE to delete *all* load case pointers (and *all* load case files, if any). Issue **LCDEF**,*LCNO*,ERASE to delete only the specific load case pointer *LCNO* (and its file, if any). With the ERASE options, all pointers are deleted; however only files with the default extension [**LCWRITE**] are deleted. Issue **LCDEF**,STAT for status of all selected load cases [**LCSEL**], or **LCDEF**,STAT,ALL for status of all load cases. The **STAT** command may be used to list all load cases. See also **LCFILE** to establish a pointer to a set of results on a load case file (written by **LCWRITE**). Harmonic element data read from a result file load case is stored at the zero-degree position.

Menu Paths

Main Menu>General Postproc>Load Case>Create Load Case
Main Menu>General Postproc>Load Case>Erase Load Case
Main Menu>General Postproc>Load Case>List Load Cases

LCFACT, *LCNO*, *FACT*

Defines scale factors for load case operations.

POST1: Load Case Calculations

MP ME ST DY <> PR EM <> <> PP ED

LCNO
Load case pointer number. If ALL, apply to all selected load cases [**LCSEL**].

FACT
Scale factor applied to load case *LCNO*. Zero (or blank) defaults to 1.0. Use a small number for a "zero" scale factor.

Command Default

All factors are 1.0.

Notes

Defines scale factors to be used in the load case operations [**LCASE** or **LCOPER**]. Scale factors are applied after an absolute value operation [**LCABS**] and are applied only to defined load cases [**LCDEF**].

Menu Paths

Main Menu>General Postproc>Load Case>Calc Options>Scale Factor

---

LCFILE, *LCNO*, *Fname*, *Ext*, --

Creates a load case from an existing load case file.

POST1: Load Case Calculations

MP ME ST DY <> PR EM <> <> PP ED

LCNO
Arbitrary (1-99) pointer number assigned to this load case.

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to the *LCNO* value preceded by an" L" (for values 10–99) or by an" L0" (for values 1–9).

- -
Unused field

Notes

Creates a load case by establishing a pointer to an existing load case file [**LCWRITE**]. This pointer (*LCNO*) can then be used on the **LCASE** or **LCOPER** commands to read the load case data into the database. This command is typically used to reestablish load case pointers in a new ANSYS session (pointers are not saved on the database file), or when more than one pointer to a single load case is desired. See the **LCDEF** command for status and erase operations. See also **LCDEF** to establish a pointer to a set of results on a results file (written during the ANSYS solution phase).

Menu Paths

Main Menu>General Postproc>Load Case>Create Load Case

LCLEAR, *NL1*, *NL2*, *NINC*

Deletes nodes and line elements associated with selected lines.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NINC*

Delete mesh for lines *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and the mesh for all selected lines [**LSEL**] is deleted. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

Notes

Deletes *all* nodes and line elements associated with selected lines (regardless of whether the nodes or elements are selected). Nodes shared by adjacent meshed lines and nodes associated with non-line elements will not be deleted. Attributes assigned as a result of **LATT** are maintained. In the program's response to the command, if a line or keypoint is tallied as "cleared," it means either its node or element reference was deleted.

Menu Paths

Main Menu>Preprocessor>Meshing>Clear>Lines

LCOMB, *NL1*, *NL2*, *KEEP*

Combines adjacent lines into one line.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1

Number of the first line to be combined. If *NL1* = ALL, *NL2* is ignored and all selected lines [**LSEL**] are combined. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* is ignored).

NL2
Number of the second line to be combined.

KEEP
Specifies whether to keep the input entities:

0
Delete lines *NL1* and *NL2* and their common keypoint. Keypoints will not be deleted if they are meshed or if they are attached to other lines. Lines will not be deleted if they are attached to different areas.

1
Keep *NL1*, *NL2*, and their common keypoint. (The common keypoint will *not* be attached to the output line.)

Notes

Combines adjacent lines into one line (the output line). This operation will effectively "undo" the **LDIV** operation. Line divisions are set to zero (use **LESIZE**, etc. to modify). Lines attached to the same area(s) can also be combined. See also the **LCCAT** command for line concatenation capability.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Add>Lines

LCOPER, *Oper*, *LCASE1*, *Oper2*, *LCASE2*
Performs load case operations.

POST1: Load Case Calculations

MP ME ST DY <> PR EM <> <> PP ED

Oper
Valid operations are:

ZERO
Zero results portion of database (*LCASE1* ignored).

SQUA
Square database values (*LCASE1* ignored).

SQRT
Square root of database (absolute) values (*LCASE1* ignored).

LPRIN
Recalculate line element principal stresses (*LCASE1* ignored). Stresses are as shown for the NMISC items of the **ETABLE** command for the specific line element type.

ADD
Add *LCASE1* to database values.

SUB
Subtract *LCASE1* from database values.

SRSS
Square root of the sum of the squares of database and *LCASE1*.

MIN
Compare and save in database the algebraic minimum of database and *LCASE1*.

MAX
Compare and save in database the algebraic maximum of database and *LCASE1*.

ABMN
Compare and save in database the absolute minimum of database and *LCASE1* (based on magnitudes, then apply the corresponding sign).

ABMX
Compare and save in database the absolute maximum of database and *LCASE1* (based on magnitudes, then apply the corresponding sign).

LCASE1
First load case in the operation (if any). See *LCNO* of the **LCDEF** command. If ALL, repeat operations using all selected load cases [**LCSEL**].

Oper2
Valid operations are:

MULT
Multiplication: *LCASE1***LCASE2*

LCASE2
Second load case. Used only with *Oper2* operations.

Notes

LCOPER operates on the database and one or two load cases according to:

Database = Database *Oper* (*LCASE1 Oper2 LCASE2*)

where operations *Oper* and *Oper2* are as described above. Absolute values and scale factors may be applied to the load cases before the operations [**LCABS**, **LCFACT**]. If *LCASE1* is not specified, only operation *Oper* is performed on the current database. If *LCASE2* is specified, operation *Oper2* will be performed *before* operation *Oper*. If *LCASE2* is not specified, operation *Oper2* is ignored. Solution items not contained [**OUTRES**] in either the database or the applicable load cases will result in a null item during a load case operation. Harmonic element data read from a result file load case are processed at zero degrees. All load case combinations are performed in the solution coordinate system, and the data resulting from load case combinations are stored in the solution coordinate system. The resultant data are then transformed to the active results coordinate system [**RSYS**] when listed or displayed.

Menu Paths

Main Menu>General Postproc>Load Case>Add
Main Menu>General Postproc>Load Case>Line Elem Stress
Main Menu>General Postproc>Load Case>Min & Max
Main Menu>General Postproc>Load Case>Square
Main Menu>General Postproc>Load Case>Square Root
Main Menu>General Postproc>Load Case>SRSS
Main Menu>General Postproc>Load Case>Subtract

LCSEL, *Type*, *LCMIN*, *LCMAX*, *LCINC*

Selects a subset of load cases.

POST1: Load Case Calculations

MP ME ST DY <> PR EM <> <> PP ED

Type
Label identifying the type of select:

S
Select a new set.

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

ALL
Restore the full set.

NONE
Unselect the full set.

INVE
Invert the current set (selected becomes unselected and vice versa).

STAT
Display the current select status.

LCMIN
Minimum value of load case pointer range.

LCMAX
Maximum value of load case pointer range. *LCMAX* defaults to *LCMIN*.

LCINC
Value increment within range. Defaults to 1. *LCINC* cannot be negative.

Command Default

All load cases are selected.

Notes

Selects a subset of load cases for other operations. For example, to select a new set of load cases based on load cases 1 through 7, use **LCSEL**,S,1,7. The subset is used when the ALL label is entered (or implied) on other commands, such as **LCFACT**, **LCABS**, **LCOPER**, etc. Load cases are flagged as selected and unselected; no load case pointers [**LCDEF**, **LCWRITE**, **LCFILE**] are actually deleted from the database.

Menu Paths

Main Menu>General Postproc>Load Case>Calc Options>Sele Ld Cases

LCSL, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Divides intersecting lines at their point(s) of intersection.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*
Numbers of lines to be intersected. If *NL1* = ALL, *NL2* to *NL9* are ignored and the intersection of all selected lines is found. If *NL1* = P, use graphical picking to specify lines (*NL2* to *NL9* are ignored).

Notes

Divides intersecting (classifies) lines at their point(s) of intersection. The original lines (and their corresponding keypoint(s)) will be deleted by default. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

This command cannot be accessed from a menu.

LCSUM, *Lab*

Specifies whether to process non-summable items in load case operations.

POST1: Results

MP ME ST DY <> PR EM <> <> PP ED

Lab
Combination option

(blank)
Only combine summable items [default].

ALL
Combine all items including non summable items.

Notes

Allows non-summable items (e.g. plastic strains) to be included in load combinations. Issue **LCSUM**,ALL before the first load case operation (**LC*XX*** command). May also be used to include nonsummable items in the appending of a results file (**RAPPND** command).

Menu Paths

This command cannot be accessed from a menu.

LCWRITE, *LCNO*, *Fname*, *Ext*, --

Creates a load case by writing results to a load case file.

POST1: Load Case Calculations

MP ME ST DY <> PR EM <> <> PP ED

LCNO
Arbitrary pointer number (1-99) to be assigned to this load case.

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to the *LCNO* value preceded by an "L" (for values 10–99) or by an "L0" (for values 1–9).

--
Unused field

Notes

Creates a load case by writing the results data in the database to a load case file. The database remains unchanged by this operation. A pointer is also established to the written set of results on the load case file. This pointer (*LCNO*) can then be used on the **LCASE** or **LCOPER** commands to read the load case data into the database. By default, only summable results data (such as displacements, stresses, elastic strains) and constant results data (such as volume) are written to the load case file unless requested (**LCSUM** command). Non-summable results data (such as plastic strains, strain energy), boundary conditions, and nodal loads are not written to the load case file. The load case file may be named by default or by a user name. Rewriting to the same file overwrites the previous data. See the **LCDEF** command for status and erase operations.

Menu Paths

Main Menu>General Postproc>Load Case>Write Load Case

LCZERO

Zeroes the results portion of the database.

POST1: Load Case Calculations

MP ME ST DY <> PR EM <> <> PP ED

Notes

Often used before the **LCOPER** command. Same as **LCOPER**,ZERO.

Menu Paths

Main Menu>General Postproc>Load Case>Zero Load Case

LDELE, *NL1*, *NL2*, *NINC*, *KSWP*

Deletes unmeshed lines.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NINC*

Delete lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and all selected lines [**LSEL**] are deleted. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

KSWP

Specifies whether keypoints are also to be deleted:

0

Delete lines only.

1

Delete lines, as well as keypoints attached to lines but not attached to other lines.

Notes

A line attached to an area cannot be deleted unless the area is first deleted.

Menu Paths

Main Menu>Preprocessor>Meshing>Concatenate>Del Concats>Lines
Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>Del Concats>Lines
Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>Del Concats>Lines
Main Menu>Preprocessor>Modeling>Delete>Del Concats>Lines
Main Menu>Preprocessor>Modeling>Delete>Line and Below
Main Menu>Preprocessor>Modeling>Delete>Lines Only
Main Menu>Preprocessor>Modeling>Topo Repair>Delete>Line and Below
Main Menu>Preprocessor>Modeling>Topo Repair>Delete>Lines Only

---

LDIV, *NL1*, *RATIO*, *PDIV*, *NDIV*, *KEEP*

Divides a single line into two or more lines.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1

Number of the line to be divided. If negative, assume *P1* (see below) is the second keypoint of the line instead of the first for *RATIO*. If ALL, divide all selected lines [**LSEL**]. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

RATIO

Ratio of line length *P1-PDIV* to line length *P1-P2*. Must be between 0.0 and 1.0. Input ignored if *NDIV* > 2.

PDIV
Number to be assigned to keypoint generated at division location (defaults to lowest available keypoint number [**NUMSTR**]). Input ignored if *NL1* = ALL or *NDIV* > 2. If *PDIV* already exists and lies on line *NL1*, divide line at *PDIV* (*RATIO* must also be 0.0). If *PDIV* already exists and does not lie on line *NL1*, *PDIV* is projected and moved to the nearest point on line *NL1* (if possible). *PDIV* cannot be attached to another line, area, or volume.

NDIV
The number of new lines to be generated from old line (defaults to 2).

KEEP
Specifies whether to keep the input entities:

0
Modify old line to use new keypoints and slopes.

1
Do not modify old line. New lines will overlay old line and have unique keypoints.

Notes

Divides a single line *NL1* (defined from keypoint *P1* to keypoint *P2*) into two or more lines. Line *NL1* becomes the new line beginning with keypoint *P1* and new lines are generated ending at keypoint *P2*. If the line is attached to an area, the area will also be updated. Line divisions are set to zero (use **LESIZE**, etc. to modify).

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line into 2 Ln's
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line into N Ln's
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Lines w/ Options

---

LDRAG, *NK1*, *NK2*, *NK3*, *NK4*, *NK5*, *NK6*, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*

Generates lines by sweeping a keypoint pattern along path.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NK1, *NK2*, *NK3*, *NK4*, *NK5*, *NK6*
List of keypoints in the pattern to be dragged (6 maximum if using keyboard entry). If *NK1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NK1* = ALL, all selected keypoints (except those that define the drag path) will be swept along the path. A component name may also be substituted for *NK1*.

NL1, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*
List of lines defining the path along which the pattern is to be dragged (6 maximum if using keyboard entry). Must be a continuous set of lines.

Notes

Generates lines (and their corresponding keypoints) by sweeping a given keypoint pattern along a characteristic drag path. If the drag path consists of multiple lines, the drag direction is determined by the sequence in which

the path lines are input (*NL1*, *NL2*, etc.). If the drag path is a single line (*NL1*), the drag direction is from the keypoint on the drag line that is closest to the first keypoint of the given pattern to the other end of the drag line.

The magnitude of the vector between the keypoints of the given pattern and the first path keypoint remains constant for all generated keypoint patterns and the path keypoints. The direction of the vector relative to the path slope also remains constant so that patterns may be swept around curves. Keypoint and line numbers are automatically assigned (beginning with the lowest available values [**NUMSTR**]). For best results, the entities to be dragged should be orthogonal to the start of the drag path. Drag operations that produce an error message may create some of the desired entities prior to terminating.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Keypoints>Along Lines

---

LDREAD, *Lab*, *LSTEP*, *SBSTEP*, *TIME*, *KIMG*, *Fname*, *Ext*, --

Reads results from the results file and applies them as loads.

SOLUTION: FE Constraints
SOLUTION: FE Body Loads

MP ME ST DY <> PR EM <> FL PP ED

Lab
Valid load label:

TEMP
Temperatures from a thermal analysis are applied as body force nodal loads [**BF**] in a structural analysis, an explicit dynamic analysis, or other type of analysis.

If the thermal analysis uses SHELL131 and/or SHELL132, temperatures are applied as body force element loads [**BFE**]. In most cases, only the top and bottom temperatures from SHELL131 and SHELL132 are used by the structural shell elements; any interior temperatures are ignored. However, for SHELL181 using section input and SHELL91, all temperatures are used. Hence, for these two cases, the number of layers must exactly match between the thermal and structural models. When using SHELL131 or SHELL132 information for the LDREAD operation, all element types should specify the same set of thermal degrees of freedom.

Can also be temperatures applied as nodal loads or initial conditions (see *KIMG* description).

FORC
Forces from a magnetic analysis are applied as force loads [**F**] in a structural analysis or a FLOTRAN analysis. **LDREAD**,FORC reads Lorentz forces in current carrying conductors and Maxwell forces on surfaces.

For a full harmonic magnetic analysis, FORC represents the time-averaged force (use in conjunction with *KIMG* = 2). Values are in the nodal coordinate system for the force loads [**F**].

HGEN
Heat generations from a magnetic analysis are applied as body force loads [**BFE**] in a thermal or a FLOTRAN analysis. For a full harmonic analysis, HGEN represents the time-averaged heat generation load (use in conjunction with *KIMG* = 2).

HFLU
Heat fluxes from a FLOTRAN analysis are applied as surface loads in a thermal analysis.

EHFLU
: Surface losses from a high frequency electromagnetic analysis are applied as time-average surface heat flux in a thermal analysis.

JS
: Source current density from a current-conduction analysis are applied as body force loads [**BFE**]. Values are in the global Cartesian coordinate system.

EF
: Electric field element centroid values from an electrostatic analysis are applied as body force loads [**BFE**] in a magnetic analysis. Values are in the global Cartesian coordinate system.

PRES
: Pressures from a FLOTRAN analysis are applied as surface loads [**SFE**] in a structural analysis. For shell elements, use the *KIMG* parameter to establish the face on which to apply the pressure.

REAC
: Reaction loads from any analysis are applied as force loads [**F**] in any analysis. Values are in the nodal coordinate system.

HFLM
: FLOTRAN predicted film coefficient (and associated bulk temperature) are applied as surface loads (film coefficient and bulk temperature) [**SFE**] in a thermal analysis. FLOTRAN film coefficients may be either positive or negative, depending on the direction of heat transfer, but ANSYS thermal analyses require positive film coefficients. If the FLOTRAN film coefficient is negative, **LDREAD** adjusts the bulk temperature to be equal to twice the wall temperature minus the bulk temperature. The adjusted bulk temperature and the now positive film coefficient make the amount and direction of heat transfer consistent with the FLOTRAN analysis.

LSTEP
: Load step number of the data set to be read. Defaults to 1. If LAST, ignore *SBSTEP* and *TIME* and read the last data set.

SBSTEP
: Substep number (within *LSTEP*). If zero (or blank), *LSTEP* represents the last substep of the load step.

TIME
: Time-point identifying the data set to be read. Used only if both *LSTEP* and *SBSTEP* are zero (or blank). If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets. If *TIME* is beyond the last time point on the file, use the last time point.

KIMG
: When used with results from harmonic analyses (**ANTYPE**,HARMIC) *KIMG* establishes which set of data to read:

  0
  : Read the real part of the solution. Valid also for *Lab* = EHFLU to read in time-average heat flux.

  1
  : Read the imaginary part of the solution.

  2
  : Calculate and read the time-average part. Meaningful for *Lab* = HGEN or FORC.

  When used with the PRES label, *KIMG* represents the shell element face on which to apply the pressure:

  1
  : Apply pressure to face 1

2
Apply pressure to face 2

When used with the TEMP label, *KIMG* indicates how temperatures are to be applied. In an explicit dynamic analysis, KIMG = 0 is the only valid option for applying temperature loads.

0
Apply temperatures as body loads

1
Apply temperatures as nodal loads

2
Apply temperatures as initial conditions.

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to RST if *Fname* is blank.

--
Unused field

Notes

Reads results data from the results file and applies them as loads. You also can use **LDREAD** to apply results from an analysis defined with one physics environment as loads on a second analysis using a different physics environment. The values are applied as loads for field coupling effects (such as output temperatures from a thermal analysis as input to a structural analysis). Nodal loads are applied only to selected nodes. Element loads are applied only to selected elements. Element surface loads are applied only to selected elements where all face nodes for that surface are selected. Additionally, to assure proper distribution of the surface loads, select *only* the nodes on the element face where the surface load is to be applied. Scaling and accumulation specifications are applied as the loads are read (**BFCUM** for body force loads, **SFCUM** for surface loads, and **FCUM** for force loads). These commands do not work for tabular boundary conditions or temperature loads applied to an explicit dynamic analysis via **LDREAD**. Use the appropriate list command to list the results (**BFLIST** or **BFELIST** for body force loads, **SFELIST** for surface loads, and **FLIST** for force loads). Values may be redefined after being read by issuing **LDREAD** again with a different load step and substep, or time value.

When you use **LDREAD** in an explicit dynamic analysis to read in temperatures, you cannot use the **EDLOAD**,,TEMP command to apply temperature loading. Furthermore, any temperature loading defined by **LDREAD** cannot be listed or deleted by the **EDLOAD** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>From Therm Analy

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>From Mag Analy
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Forces>Forces>From Mag Analy
Main Menu>Preprocessor>Loads>Define Loads>Apply>Initial Condit'n>Temp from ANSYS
Main Menu>Preprocessor>Loads>Define Loads>Apply>Initial Condit'n>Temp from Fluid
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>From Therm Analy
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>From Elec An
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>Electric Field>From Elec An
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>From Mag Analy
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>From Reactions
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>From Fluid Analy
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Temperature>From Therm Analy
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>From Fluid Analy
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>From EMAG Analy
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>From Fluid Analy
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>From Mag Analy
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>From ANSYS
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>From Flotran
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Temp From ANSYS
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>From Therm Analy
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>From Mag Analy
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Forces>Forces>From Mag Analy
Main Menu>Solution>Define Loads>Apply>Initial Condit'n>Temp from ANSYS
Main Menu>Solution>Define Loads>Apply>Initial Condit'n>Temp from Fluid
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>From Therm Analy
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>From Elec An
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>Electric Field>From Elec An
Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>From Mag Analy
Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>From Reactions
Main Menu>Solution>Define Loads>Apply>Structural>Pressure>From Fluid Analy
Main Menu>Solution>Define Loads>Apply>Structural>Temperature>From Therm Analy
Main Menu>Solution>Define Loads>Apply>Thermal>Convection>From Fluid Analy
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>From EMAG Analy
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>From Fluid Analy
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>From Mag Analy
Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>From ANSYS
Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>From Flotran
Main Menu>Solution>Loading Options>Temp From ANSYS

LESIZE, *NL1*, *SIZE*, *ANGSIZ*, *NDIV*, *SPACE*, *KFORC*, *LAYER1*, *LAYER2*, *KYNDIV*

Specifies the divisions and spacing ratio on unmeshed lines.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NL1

Number of the line to be modified. If ALL, modify all selected lines [**LSEL**]. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

SIZE

If *NDIV* is blank, *SIZE* is the division (element edge) length. The number of divisions is automatically calculated from the line length (rounded upward to next integer). If *SIZE* is zero (or blank), use *ANGSIZ* or *NDIV*.

ANGSIZ

The division arc (in degrees) spanned by the element edge (except for straight lines, which always result in one division). The number of divisions is automatically calculated from the line length (rounded upward to next integer).

NDIV

If positive, *NDIV* is the number of element divisions per line. If -1 (and *KFORC* = 1), *NDIV* is assumed to be zero element divisions per line. TARGE169 with a rigid specification ignores *NDIV* and will always mesh with one element division.

SPACE

Spacing ratio. If positive, nominal ratio of last division size to first division size (if > 1.0, sizes increase, if < 1.0, sizes decrease). If negative, |*SPACE*| is nominal ratio of center division(s) size to end divisions size. Ratio defaults to 1.0 (uniform spacing). For layer-meshing, a value of 1.0 normally is used. If *SPACE* = FREE, ratio is determined by other considerations

KFORC

KFORC 0-3 are used only with *NL1* = ALL. Specifies which selected lines are to be modified.

0

Modify only selected lines having undefined (zero) divisions.

1

Modify all selected lines.

2

Modify only selected lines having fewer divisions (including zero) than specified with this command.

3

Modify only selected lines having more divisions than specified with this command.

4

Modify only nonzero settings for *SIZE*, *ANGSIZ*, *NDIV*, *SPACE*, *LAYER1*, and *LAYER2*. If *KFORC* = 4, blank or 0 settings remain unchanged.

LAYER1

Layer-meshing control parameter. Distance which defines the thickness of the inner mesh layer, *LAYER1*. Elements in this layer are uniformly-sized with edge lengths equal to the specified element size for the line (either through *SIZE* or line-length/*NDIV*). A positive value for *LAYER1* is interpreted as an absolute length, while a negative value in interpreted as a multiplier on the specified element size for the line. As a general rule, the resulting thickness of the inner mesh layer should be greater than or equal to the specified element size for the line. If *LAYER1* = OFF, layer-meshing control settings are cleared for the selected lines. The default value is 0.0

LAYER2

Layer-meshing control parameter. Distance which defines the thickness of the outer mesh layer, *LAYER2*. Elements in this layer transition in size from those in *LAYER1* to the global element size. A positive value of *LAYER2* is interpreted as an absolute length, while a negative value is interpreted as a mesh transition factor. A value of *LAYER2* = -2 would indicate that elements should approximately double in size as the mesh progresses normal to *LAYER1*. The default value is 0.0.

KYNDIV

0, No, and Off means that SmartSizing cannot override specified divisions and spacing ratios. Mapped mesh fails if divisions do not match. This defines the specification as “hard”.

1, Yes, and On means that SmartSizing can override specified divisions and spacing ratios for curvature or proximity. Mapped meshing can override divisions to obtain required matching divisions. This defines the specification as“ soft”.

Notes

Defines the number of divisions and the spacing ratio on selected lines. Lines with previously specified divisions may also be changed.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Layers>Clr Layers
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Layers>Picked Lines
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>All Lines
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Clr Size
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Copy Divs
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Flip Bias
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Picked Lines

---

LEXTND, *NL1*, *NK1*, *DIST*, *KEEP*

Extends a line at one end by using its slope.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1

Number of the line to be extended. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NK1

Number of keypoint at the end of line *NL1* to be extended.

DIST

Distance that the line will be extended.

KEEP

Specifies whether to keep the input entities:

0

Modify old line to use new keypoints and slopes.

1

Do not modify old line. New line will overlay old line and have unique keypoints.

Notes

Extends a line at one end by using its slope. Lines may be redefined only if not yet attached to an area. Line divisions are set to zero (use **LESIZE**, etc. to modify). Note that solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extend Line

LFILLT, *NL1*, *NL2*, *RAD*, *PCENT*

Generates a fillet line between two intersecting lines.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1
Number of the first intersecting line. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NL2
Number of the second intersecting line.

RAD
Radius of fillet to be generated. Radius should be less than the lengths of the two lines specified with *NL1* and *NL2*.

PCENT
Number to be assigned to generated keypoint at fillet arc center. If zero (or blank), no keypoint is generated.

Notes

Generates a fillet line between two intersecting lines *NL1* (*P1-PINT*) and *NL2* (*P2-PINT*). Three keypoints may be generated, two at the fillet tangent points (*PTAN1* and *PTAN2*) and one (optional) at the fillet arc center (*PCENT*). Line *P1-PINT* becomes *P1-PTAN1*, *P2-PINT* becomes *P2-PTAN2*, and new arc line *PTAN1-PTAN2* is generated. Generated keypoint and line numbers are automatically assigned (beginning with the lowest available values [**NUMSTR**]). Line divisions are set to zero (use **LESIZE**, etc. to modify).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Line Fillet

LFSURF, *SLINE*, *TLINE*

Generates surface elements overlaid on the edge of existing solid elements and assigns the extra node as the closest fluid element node.

PREP7: Elements

MP ME <> <> <> PR <> <> <> <> ED

SLINE
Component name for the surface lines of the meshed solid areas.

TLINE
Component name for the target lines meshed with fluid elements.

Notes

This command macro is used to generate surface effect elements overlaid on the surface of existing plane elements and, based on proximity, to determine and assign the extra node for each surface element. The underlying areas of the solid region and the fluid lines must be meshed prior to calling this command macro.

The surface lines of the solid and the target lines of the fluid are grouped into components and named using the **CM** command. The names must be enclosed in single quotes (e.g., 'SLINE') when the **LFSURF** command is manually typed in.

When using the GUI method, node and element components are created through the picking dialog boxes associated with this command.

The macro is applicable for the SURF151 and FLUID116 element types.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Area to Fluid
Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Line to Fluid
Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Node to Fluid

---

LGEN, *ITIME*, *NL1*, *NL2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

Generates additional lines from a pattern of lines.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

ITIME

Do this generation operation a total of *ITIME*s, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be > 1 for generation to occur.

NL1, *NL2*, *NINC*

Generate lines from pattern beginning with *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and pattern is all selected lines [**LSEL**]. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

DX, *DY*, *DZ*

Keypoint location increments in the active coordinate system (--, Dθ, DZ for cylindrical, --, Dθ, -- for spherical).

KINC

Keypoint increment between generated sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM

Specifies if elements and nodes are also to be generated:

0

Generate nodes and elements associated with the original lines, if they exist.

1
Do not generate nodes and elements.

IMOVE
Specifies whether to redefine the existing lines:

0
Generate additional lines as requested with the *ITIME* argument.

1
Move original lines to new position retaining the same keypoint numbers (*ITIME*, *KINC*, and *NOELM* are ignored). Valid only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates additional lines (and their corresponding keypoints and mesh) from a given line pattern. The MAT, TYPE, REAL, and ESYS attributes are based upon the lines in the pattern and not upon the current settings. End slopes of the generated lines remain the same (in the active coordinate system) as those of the given pattern. For example, radial slopes remain radial, etc. Generations which produce lines of a size or shape different from the pattern (i.e., radial generations in cylindrical systems, radial and phi generations in spherical systems, and theta generations in elliptical systems) are not allowed. Note that solid modeling in a toroidal coordinate system is not recommended. New line numbers are automatically assigned (beginning with the lowest available values [**NUMSTR**]).

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Lines
Main Menu>Preprocessor>Modeling>Move / Modify>Lines

LGLUE, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Generates new lines by "gluing" lines.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*
Numbers of the lines to be glued. If *NL1* = ALL, all selected lines will be glued (*NL2* to *NL9* will be ignored). If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

Notes

Use of the **LGLUE** command generates new lines by "gluing" input lines. The glue operation redefines the input lines so that they share keypoints at their common ends. The new lines encompass the same geometry as the original lines. This operation is only valid if the intersections of the input lines are keypoints at the ends of those lines. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

The **LGLUE** command results in the merging of keypoints at the common end of the lines. The keypoints of the lower numbered line will be kept. This means one must be aware of line numbering when multiple **LGLUE** commands are applied to avoid any "ungluing" of geometry.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Glue>Lines

LGWRITE, *Fname*, *Ext*, --, *Kedit*

Writes the database command log to a file.

SESSION: Files

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to LGW if *Fname* and *Ext* are blank.

--
Unused field

Kedit
Flag to suppress nonessential commands:

NONE
Do not suppress any commands (default).

COMMENT
Write nonessential commands as comments (starting with !).

REMOVE
Do not write nonessential commands or comments.

Notes

Writes the database command log to a named file. The database command log contains all commands that were used to create the current database. These commands are recorded in the database as they are issued, and saved in the database file (**File.DB**) whenever the database is saved. The **LGWRITE** command extracts these commands from the database and writes them to a file. Nonessential commands (for listing, graphics displays, help, etc.) can be excluded from the file by using the *Kedit* field. The file resulting from **LGWRITE** can be used as command input to the program. This command is most useful if the session log file (**File.LOG**), which is normally saved during an interactive session, has been lost or corrupted. **LGWRITE** cannot be used after design optimization looping because the database is cleared [**/CLEAR**] automatically at each loop; use **File.LOG** instead.

This command is valid in any processor.

Menu Paths

Utility Menu>File>Write DB Log File

/LIGHT, *WN*, *NUM*, *INT*, *XV*, *YV*, *ZV*, *REFL*

Specifies the light direction for the display window.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

NUM
Ambient or directional light key:

0
Ambient light (default).

1
Directional light.

INT
Light intensity factor (defaults to 0.3 for ambient, 1.0 for directional). This option is valid only for 3-D devices).

XV, *YV*, *ZV*
Light direction (valid only for *NUM* = 1). The directional light source is parallel to the line from point *XV*, *YV*, *ZV* to the origin, in the global Cartesian system origin. Defaults to the viewing direction [**/VIEW**].

REFL
Light reflectance factor (valid only for *NUM* = 1 and 3-D devices).

Command Default

Use ambient light.

Notes

Defines the light direction for the window. Use this command only with 3-D graphics devices or 2-D devices when Z-buffering is used [**/TYPE**,,(6 or 7)]. The ambient light has no direction, only an intensity. You can position the directional light source by defining a point (in the global Cartesian coordinate system) representing a point along the light directional line. This point, and the global Cartesian coordinate system origin, define the line along which the light is positioned looking toward the origin. You can use any point along the light line; for example, both (1.,1.,1.) and (2.,2.,2.) give the same light effect. For 3-D graphics devices only, the directional light source also has intensity and reflectance factors.

By choosing the highest intensity ambient light for 3-D graphics devices (via the command **/LIGHT**,WN,0,1), you can nullify color shading and other effects of directional lighting.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Light Source

LINA, *NL*, *NA*

Finds the intersection of a line with an area.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL

Number of line to be intersected. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NA

Number of area to be intersected.

Notes

Finds the intersection of a line with an area. New lines will be generated where the lines intersect the areas. If the regions of intersection are only points, new keypoints will be generated instead. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Line with Area

---

LINE

Specifies "Lines" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

This command cannot be accessed from a menu.

/LINE, *X1*, *Y1*, *X2*, *Y2*

Creates annotation lines (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

X1
Line X starting location (-1.0 < X < 2.0).

Y1
Line Y starting location (-1.0 < Y < 1.0).

X2
Line X ending location (-1.0 < X < 2.0).

Y2
Line Y ending location (-1.0 < Y < 1.0).

Notes

Defines annotation lines to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All lines are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** command to set the attributes of the line.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

LINES, *N*

Specifies the length of a printed page.

POST26: Listing

MP ME ST DY <> PR EM <> FL PP ED

N
Number of lines per page (defaults to 20). (Minimum allowed = 11).

Command Default

20 lines per page.

Notes

Specifies the length of a printed page (for use in reports, etc.).

Menu Paths

Main Menu>TimeHist Postpro>Settings>List

LINL, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Finds the common intersection of lines.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*
Numbers of lines to be intersected. If *NL1* = ALL, find the intersection of all selected lines and *NL2* to *NL9* are ignored. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

Notes

Finds the common (not pairwise) intersection of lines. The common intersection is defined as the regions shared (in common) by *all* lines listed on this command. New lines will be generated where the original lines intersect. If the regions of intersection are only points, new keypoints will be generated instead. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Common>Lines

---

LINP, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Finds the pairwise intersection of lines.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*
Numbers of lines to be intersected pairwise. If *NL1* = ALL, find the pairwise intersection of all selected lines and *NL2* to *NL9* are ignored. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NL1*.

Notes

Finds the pairwise intersection of lines. The pairwise intersection is defined as any and all regions shared by at least two lines listed on this command. New lines will be generated where the original lines intersect pairwise. If the regions of pairwise intersection are only points, new keypoints will be generated. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Pairwise>Lines

---

LINV, *NL*, *NV*

Finds the intersection of a line with a volume.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL
Number of line to be intersected. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NV
Number of volume to be intersected.

Notes

Finds the intersection of a line with a volume. New lines will be generated where the lines intersect the volumes. If the regions of intersection are only points, new keypoints will be generated instead. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Line with Volume

---

LIST, *LEVEL*

Lists out the sets in the results file.

AUX3: Results Files

MP ME ST DY <> PR EM <> FL PP ED

Notes

This command lists the results set number, the load step, substep, and time step for each set. It also shows all sets marked for deletion.

Menu Paths

This command cannot be accessed from a menu.

***LIST**, *Fname*, *Ext*, --

Displays the contents of an external, coded file.

SESSION: Files

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Ext
Filename extension (8 character maximum).

--
Unused field

Notes

Displays the contents of an external, coded file. The file to be listed cannot be in use (open) at the time (except for the error file, **File.ERR**, which may be displayed with ***LIST**,ERR).

Use caution when you are listing active ANSYS files via the **List> Files> Other** and **File> List> Other** menu paths. File I/O buffer and system configurations can result in incomplete listings unless the files are closed.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

LLIST, *NL1*, *NL2*, *NINC*, *Lab*

Lists the defined lines.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NINC*
List lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL (default), *NL2* and *NINC* are ignored and all selected lines [**LSEL**] are listed. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

Lab
Determines what type of listing is used (one of the following):

(blank)
Prints information about all lines in the specified range.

RADIUS
Prints the radius of certain circular arcs, along with the keypoint numbers of each line. Straight lines, non-circular curves, and circular arcs not internally identified as arcs (which depends upon how each arc is created) will print a radius value of zero.

LAYER
Prints layer-mesh control specifications.

HPT
Prints information about only those lines that contain hard points. HPT is not supported in the GUI.

ORIENT
Prints a list of lines, and identifies any orientation keypoints and any cross section IDs that are associated with the lines. Used for beam meshing with defined orientation nodes and cross sections.

Notes

There are 2 listings for the number of element divisions and the spacing ratio. The first listing shows assignments from **LESIZE** only, followed by the "hard" key (*KYNDIV*). See **LESIZE** for more information. The second listing shows *NDIV* and *SPACE* for the existing mesh, if one exists. Whether this existing mesh and the mesh generated by **LESIZE** match at any given point depends upon meshing options and the sequence of meshing operations.

A "-1" in the "nodes" column indicates that the line has been meshed but that there are no interior nodes.

An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **LATT** command (and will not be reset to zero if the mesh is cleared); one listed as a negative value indicates that the attribute was assigned using the attribute pointer [TYPE, MAT, REAL, or ESYS] that was active during meshing (and will be reset to zero if the mesh is cleared).

This command is valid in any processor.

Menu Paths

Utility Menu>List>Lines

LMATRIX, *SYMFAC*, *Coilname*, *Curname*, *Indname*

Calculates an inductance matrix and the total flux linkage for an N-winding coil system.

SOLUTION: Analysis Options

MP ME ST <> <> <> EM <> <> PP ED

SYMFAC
Geometric symmetry factor. Inductance terms are scaled by this factor which represents the fraction of the total device modeled. Default is 1.

Coilname
Alphanumeric prefix identifier for coil label used in defining named element coil components. Default is 'coil.'

Curname
Name of a predefined parameter array containing the nominal coil currents of the system. The array must be defined (see ***DIM** command) prior to calling the **LMATRIX** macro. Default is 'cur.'

Indname
Name of the array parameter to be created by **LMATRIX** containing the calculated inductance matrix and the flux linkage in each coil. A text file of the same name with an extension **.TXT** is created containing the matrix data. Default is 'ind.'

Notes

LMATRIX calculates the differential inductance matrix for an *N*-winding system where *N* is the number of coils in the system, and calculates the total flux linkage in each coil. **LMATRIX** may only be executed after the solution of a problem with nominal currents applied to the coils at a desired "operating point." The array *Indname* has *N* rows and *N*+1 columns. The *N* x *N* block is the differential inductance matrix; the *N*+1th column contains the total flux linkage, with the *i*th row corresponding to the *i*th coil. See the *ANSYS, Inc. Theory Reference* for more details.

To invoke the **LMATRIX** macro, for the classical formulations, the elements for each coil must be grouped into a component using the **CM** command. Each set of independent coil elements is assigned a component name with the prefix *Coilname* followed by the coil number. For the solenoidal formulations, you must make the exciting node with a **F**,AMPS load a node component using the **CM** command. The classical and solenoidal formulations cannot be mixed.

To invoke the **LMATRIX** macro, the vector array parameter *Curname* with dimension *N* must be defined and named using the ***DIM** command. You must set each vector array entry equal to the nominal current per turn in the corresponding coil at the operating point. Zero current values must be approximated by a negligibly small applied current.

Do not apply (or remove) inhomogeneous loads before using the **LMATRIX** command. Inhomogeneous loads are those created by:

- Degree of freedom commands (**D**, **DA**, etc.) specifying nonzero degrees of freedom values on nodes or solid model entities
- Any **CE** command with a nonzero constant term

Do not put any loads (for example, current) on elements not contained in the element component.

Operating solutions must be obtained through static analysis before calling **LMATRIX**. All name-strings must be enclosed in single quotes in the **LMATRIX** command line. The geometric symmetry factor, *Symfac*, represents the fraction of the device modeled, disregarding any current source primitives.

LMATRIX works with the SOLID97 and SOLID117 solenoidal formulations. For more information, see LMATRIX in the *ANSYS Low-Frequency Electromagnetic Analysis Guide*.

See the *ANSYS, Inc. Theory Reference* and Electric and Magnetic Macros in the *ANSYS Low-Frequency Electromagnetic Analysis Guide* for details.

Menu Paths

Main Menu>Solution>Solve>Electromagnet>Static Analysis>Induct Matrix

---

LMESH, *NL1*, *NL2*, *NINC*

Generates nodes and line elements along lines.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NINC*

Mesh lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and all selected lines [**LSEL**] are meshed. If *NL1* = P, graphical picking is enabled and all remaining

command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

Notes

Generates nodes and line elements along lines. Missing nodes required for the generated elements are created and assigned the lowest available numbers.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Lines

LNCOLLAPSE, *LINE*, *KEYPOINT*

Collapse a line segment to a keypoint (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

LINE
The ID of the line segment to be collapsed.

KEYPOINT
The ID of the existing keypoint (one of the keypoints connected to the line segment) that will remain after the collapse operation.

Notes

This command is available only for repairing the geometry of models imported from CAD systems (Default IGES option).

This command does not check for any entity penetration resulting from the collapse.

Menu Paths

Main Menu>Preprocessor>Modeling>Simplify>Toolkit>Collapse Lines

---

LNDETACH, *LINE1*, *LINE2*, *LNINC*

Detaches lines from neighboring geometric entity (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

LINE1, *LINE2*, *LNINC*
Detach lines from *LINE1* to *LINE2* (defaults to *LINE1*) in steps of *LNINC* (defaults to 1).

Notes

Use this command to detach non-manifold lines from their neighboring geometric entities. This command is available only for repairing the geometry of models imported from CAD systems.

Menu Paths

Main Menu>Preprocessor>Modeling>Geom Repair>Detach Lines

LNFILL, *KP1*, *KP2*

Creates a straight line between two keypoints (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

KP1, *KP2*

Creates a straight line from keypoint defined by *KP1* to keypoint defined by *KP2*. If *KP1* = P; graphical picking is enabled and the remaining command field is ignored.

Notes

This command is available only for repairing the geometry of models imported from CAD systems (Default IGES option).

Menu Paths

Main Menu>Preprocessor>Modeling>Geom Repair>Fill Lines

LNMERGE, *LN1*, *LN2*, *LN3*, *LN4*, *LN4*, *LN6*, *LN7*, *LN8*, *LN9*, *LN10*

Merges two or more connected line segments (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

LN1, *LN2*, *LN3*, *LN4*, *LN4*, *LN6*, *LN7*, *LN8*, *LN9*, *LN10*

List of lines that define the set of line segments to merge into one or more line segments. The command merges all the lines from the selected set that are connected to only one other line. If *LN1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). If *LN1* = ALL, all selected lines will be merged and all remaining arguments are ignored. The connected lines are automatically grouped together to form a set of merged lines.

Notes

Use the **LNMERGE** command to merge disproportionately small line segments into a single line segment. This command is available only for repairing the geometry of models imported from CAD systems (Default IGES option). Only those line segments specified in the line list that are connected together will be joined by the merge operation. The command considers two lines to be connected if those two lines (and only those two lines) connect together at a single keypoint.

While the **LNMERGE** command allows you to join multiple lines in one merge operation, the preferred method is to merge pairs of line segments to successively build a single line segment. Using this technique can prevent problems during meshing.

The **LNMERGE** command does not check the angle of inclination between adjacent lines during the merge operation.

Menu Paths

Main Menu>Preprocessor>Modeling>Simplify>Toolkit>Merge Lines

LNSPLIT, *LINE*, *PARAM*

Splits a line segment into two line segments (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

LINE
ID identifying the line to be split.

PARAM
Arc length parameter specifying the location of the keypoint which will join the two new line segments (the "split point"). Valid arguments are values between 0 and 1, measured from the line's starting keypoint. Negative values between 0 and -1 are measured from the line's ending keypoint.

Notes

This command is available only for repairing the geometry of models imported from CAD systems (Default IGES option).

Menu Paths

Main Menu>Preprocessor>Modeling>Simplify>Toolkit>Split Lines

LNSRCH, *Key*

Activates a line search to be used with Newton-Raphson.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR <> <> <> PP ED

Key
Line search key:

OFF
Do not use a line search.

ON
Use a line search. Note, adaptive descent is suppressed when **LNSRCH** is on unless explicitly requested on the **NROPT** command. *Having line search on and adaptive descent on at the same time is not recommended.*

AUTO
ANSYS automatically switches line searching ON and OFF between substeps of a load step as needed. This option is recommended.

Command Default

Line search off, unless contact elements are present.

Notes

The default values given for this command assume **SOLCONTROL**,ON (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL**,ON and **SOLCONTROL**,OFF.

Activates a line search to be used with the Newton-Raphson method [**NROPT**]. Line search is an alternative to adaptive descent (see the *ANSYS, Inc. Theory Reference*).

LNSRCH,AUTO can be very efficient for problems in which **LNSRCH** is needed at only certain substeps.

You cannot use line search [**LNSRCH**], automatic time stepping [**AUTOTS**], or the DOF solution predictor [**PRED**] with the arc-length method [**ARCLEN**, **ARCTRM**]. If you activate the arc-length method after you set **LNSRCH**, **AUTOTS**, or **PRED**, a warning message appears. If you choose to proceed with the arc-length method activation, ANSYS disables your line search, automatic time stepping, and DOF predictor settings.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Line Search
Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Solution>Load Step Opts>Nonlinear>Line Search

LOCAL, *KCN*, *KCS*, *XC*, *YC*, *ZC*, *THXY*, *THYZ*, *THZX*, *PAR1*, *PAR2*

Defines a local coordinate system by a location and orientation.

DATABASE: Coordinate System

MP ME ST DY <> PR EM <> FL PP ED

KCN
Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

KCS
Coordinate system type:

0 or CART
Cartesian

1 or CYLIN
Cylindrical (circular or elliptical)

2 or SPHE
Spherical (or spheroidal)

3 or TORO
Toroidal

XC, *YC*, *ZC*
Location (in the global Cartesian coordinate system) of the origin of the new coordinate system.

THXY
First rotation about local Z (positive X toward Y).

THYZ
Second rotation about local X (positive Y toward Z).

THZX
Third rotation about local Y (positive Z toward X).

PAR1
Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

PAR2
Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

Notes

Defines a local coordinate system by origin location and orientation angles. The local coordinate system is parallel to the global Cartesian system unless rotated. Rotation angles are in degrees and redefine any previous rotation angles. See the **CLOCAL**, **CS**, **CSWPLA**, and **CSKP** commands for alternate definitions. This local system becomes the active coordinate system [**CSYS**]. Local coordinate systems may be displayed with the **/PSYMB** command.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>At Specified Loc

---

LOVLAP, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Overlaps lines.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*
Numbers of lines to be overlapped. If *NL1* = ALL, *NL2* to *NL9* are ignored and all selected lines are overlapped. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

Notes

Overlaps lines. Generates new lines which encompass the geometry of all the input lines. The new lines are defined by the regions of intersection of the input lines, and by the complementary (non-intersecting) regions. See the *ANSYS Modeling and Meshing Guide* for an illustration. This operation is only valid when the region of intersection is a line. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Overlap>Lines

---

LPLOT, *NL1*, *NL2*, *NINC*

Displays the selected lines.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NINC*

Display lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL (default), *NL2* and *NINC* are ignored and display all selected lines [**LSEL**].

Notes

Mesh divisions on plotted lines are controlled by the *LDIV* option of the **/PSYMB** command.

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Lines
Utility Menu>Plot>Specified Entities>Lines

LPTN, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Partitions lines.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Numbers of lines to be operated on. If *NL1* = ALL, *NL2* to *NL9* are ignored all selected lines are used. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NL1*.

Notes

Partitions lines. Generates new lines which encompass the geometry of all the input lines. The new lines are defined by both the regions of intersection of the input lines and the complementary (non-intersecting) regions. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Partition>Lines

---

LREFINE, *NL1*, *NL2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

Refines the mesh around specified lines.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NINC*

Lines (*NL1* to *NL2* in increments of *NINC*) around which the mesh is to be refined. *NL2* defaults to *NL1*, and *NINC* defaults to 1. If *NL1* = ALL, *NL2* and *NINC* are ignored and all selected lines are used for refinement. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

LEVEL

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

DEPTH

Depth of mesh refinement in terms of the number of elements outward from the indicated lines (defaults to 1).

POST

Type of postprocessing to be done after element splitting, in order to improve element quality:

OFF

No postprocessing will be done.

SMOOTH

Smoothing will be done. Node locations may change.

CLEAN

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

RETAIN

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The ANSYS program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

ON

The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

OFF

The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

Notes

LREFINE performs local mesh refinement around the specified lines. By default, the indicated elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

LREFINE refines all area elements and tetrahedral volume elements that are adjacent to the specified lines. Any volume elements that are adjacent to the specified lines, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes [**IC**], coupled nodes [**CP** family of commands], constraint equations [**CE** family of commands], or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. For additional restrictions on mesh refinement, see Revising Your Model in the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Lines

LREVERSE, *LNUM*, *NOEFLIP*

Reverses the normal of a line, regardless of its connectivity or mesh status.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

LNUM

Line number of the line whose normal direction is to be reversed. If *LNUM* = ALL, the normals of all selected lines will be reversed. If *LNUM* = P, graphical picking is enabled. A component name may also be substituted for *LNUM*.

NOEFLIP

Indicates whether you want to change the normal direction of the existing elements on the reversed line(s) so that they are consistent with each line's new normal direction.

0

Make the normal direction of existing elements on the reversed line(s) consistent with each line's new normal direction (default).

1

Do not change the normal direction of existing elements on the reversed line(s).

Notes

You cannot use the **LREVERSE** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal.

For more information, see Revising Your Model in the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Reverse Normals>of Lines

---

LROTAT, *NK1*, *NK2*, *NK3*, *NK4*, *NK5*, *NK6*, *PAX1*, *PAX2*, *ARC*, *NSEG*

Generates circular lines by rotating a keypoint pattern about an axis.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NK1, *NK2*, *NK3*, *NK4*, *NK5*, *NK6*

List of keypoints in the pattern to be rotated (6 maximum if using keyboard entry). If *NK1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NK1* = ALL, all selected keypoints (except *PAX1* and *PAX2*) will define the pattern to be rotated. A component name may also be substituted for *NK1*.

PAX1, *PAX2*

Keypoints defining the axis about which the keypoint pattern is to be rotated.

ARC

Arc length (in degrees). Positive follows right-hand rule about *PAX1-PAX2* vector. Defaults to 360.

NSEG

Number of lines (8 maximum) around circumference. Defaults to minimum required for 90° (maximum) arcs, i.e., 4 for 360°, 3 for 270°, etc.

Notes

Generates circular lines (and their corresponding keypoints) by rotating a keypoint pattern about an axis. Keypoint patterns are generated at regular angular locations (based on a maximum spacing of 90°). Line patterns are generated at the keypoint patterns. Keypoint and line numbers are automatically assigned (beginning with the lowest available values [**NUMSTR**]).

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Keypoints>About Axis

---

LSBA, *NL*, *NA*, *SEPO*, *KEEPL*, *KEEPA*

Subtracts areas from lines.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL

Line (or lines, if picking is used) to be subtracted from. If ALL, use all selected lines. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL*.

NA

Area (or areas, if picking is used) to be subtracted. If ALL, use all selected areas. A component name may also be substituted for *NA*.

SEPO

Behavior if the intersection of the lines and the areas is a keypoint or keypoints:

(blank)

The resulting lines will share keypoint(s) where they touch.

SEPO
The resulting lines will have separate, but coincident keypoint(s) where they touch.

KEEPL
Specifies whether *NL* lines are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NL* lines after **LSBA** operation (override **BOPTN** command settings).

KEEP
Keep *NL* lines after **LSBA** operation (override **BOPTN** command settings).

KEEPA
Specifies whether *NA* areas are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete areas after **LSBA** operation (override **BOPTN** command settings).

KEEP
Keep areas after **LSBA** operation (override **BOPTN** command settings).

Notes

Generates new lines by subtracting the regions common to both *NL* lines and *NA* areas (the intersection) from the *NL* lines. The intersection can be a line(s) or keypoint(s). If the intersection is a keypoint and *SEP0* is blank, the *NL* line is divided at the keypoint and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEP0* is set to SEPO, *NL* is divided into two unconnected lines with separate keypoints where they touch. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by Area
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by Area

---

LSBL, *NL1*, *NL2*, *SEPO*, *KEEP1*, *KEEP2*

Subtracts lines from lines.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL1
Line (or lines, if picking is used) to be subtracted from. If ALL, use all selected lines. Lines specified in this argument are not available for use in the *NL2* argument. If P, graphical picking is enabled (valid only in the GUI) and all remaining fields are ignored. A component name may also be substituted for *NL1*.

NL2
Line (or lines, if picking is used) to subtract. If ALL, use all selected lines (except those included in the *NL1* argument). A component name may also be substituted for *NL2*.

SEPO
Behavior if the intersection of the *NL1* lines and the *NL2* lines is a keypoint or keypoints:

(blank)
The resulting lines will share keypoint(s) where they touch.

SEPO
The resulting lines will have separate, but coincident keypoint(s) where they touch.

KEEP1
Specifies whether *NL1* lines are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NL1* lines after **LSBL** operation (override **BOPTN** command settings).

KEEP
Keep *NL1* lines after **LSBL** operation (override **BOPTN** command settings).

KEEP2
Specifies whether *NL2* lines are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NL2* lines after **LSBL** operation (override **BOPTN** command settings).

KEEP
Keep *NL2* lines after **LSBL** operation (override **BOPTN** command settings).

Notes

Generates new lines by subtracting the regions common to both *NL1* and *NL2* lines (the intersection) from the *NL1* lines. The intersection can be a line(s) or point(s). If the intersection is a point and *SEPO* is blank, the *NL1* line is divided at the point and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEPO* is set to SEPO, *NL1* is divided into two unconnected lines with separate keypoints where they touch. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **LSBL**,ALL,ALL will have no effect since all the lines (in *NL1*) will be unavailable as *NL2* lines.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by Line
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by Line
Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Lines
Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>With Options>Lines

LSBV, *NL*, *NV*, *SEPO*, *KEEPL*, *KEEPV*

Subtracts volumes from lines.

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL

Line (or lines, if picking is used) to be subtracted from. If ALL, use all selected lines. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL*.

NV

Volume (or volumes, if picking is used) to be subtracted. If ALL, use all selected volumes. A component name may also be substituted for *NV*.

SEPO

Behavior if the intersection of the *NL* lines and the *NV* volumes is a keypoint or keypoints:

(blank)
The resulting lines will share keypoint(s) where they touch.

SEPO
The resulting lines will have separate, but coincident keypoint(s) where they touch.

KEEPL

Specifies whether *NL* lines are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NL* lines after **LSBV** operation (override **BOPTN** command settings).

KEEP
Keep *NL* lines after **LSBV** operation (override **BOPTN** command settings).

KEEPV

Specifies whether *NV* volumes are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NV* volumes after **LSBV** operation (override **BOPTN** command settings).

KEEP
Keep *NV* volumes after **LSBV** operation (override **BOPTN** command settings).

Notes

Generates new lines by subtracting the regions common to both *NL* lines and *NV* volumes (the intersection) from the *NL* lines. The intersection can be a line(s) or point(s). If the intersection is a point and *SEPO* is blank, the *NL1* line is divided at the point and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEPO* is set to SEPO, *NL1* is divided into two unconnected lines with separate keypoints where they touch. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned

to the original entities will not be transferred to the new entities generated. **LSBL**,ALL,ALL will have no effect since all the lines (in *NL1*) will be unavailable as *NL2* lines.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by Volume
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by Volume
Main Menu>Preprocessor>Modeling>Operate>Divide>Line by Volu

LSBW, *NL*, *SEPO*, *KEEP*

Subtracts the intersection of the working plane from lines (divides lines).

PREP7: Booleans

MP ME ST DY <> PR EM <> FL PP ED

NL
Line (or lines, if picking is used) to be subtracted from. If *NL* = ALL, use all selected lines. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be input for *NL*.

SEPO
Behavior of the created boundary.

(blank)
The resulting lines will share keypoint(s) where they touch.

SEPO
The resulting lines will have separate, but coincident keypoint(s).

KEEP
Specifies whether *NL* lines are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NL* lines after **LSBW** operation (override **BOPTN** command settings).

KEEP
Keep *NL* lines after **LSBW** operation (override **BOPTN** command settings).

Notes

Generates new lines by subtracting the intersection of the working plane from the *NL* lines. The intersection will be a keypoint(s). The working plane must not be in the same plane as the *NL* line(s). If *SEPO* is blank, the *NL* line is divided and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEPO* is set to SEPO, *NL* is divided into two unconnected lines with separate keypoints. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. Areas that completely contain the input lines will be updated if the lines are divided by this operation.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by WrkPlane
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by WrkPlane
Main Menu>Preprocessor>Modeling>Operate>Divide>Line by WrkPlane

---

LSCLEAR, *Lab*

Clears loads and load step options from the database.

SOLUTION: Load Step Operations

MP ME ST <> <> PR EM <> FL PP ED

Lab
Label identifying the data to be cleared:

SOLID
Delete only solid model loads.

FE
Delete only finite element loads.

INER
Delete only inertia loads (**ACEL**, etc.).

LFACT
Initialize only load factors (on **DCUM**, **FCUM**, **SFCUM**, etc.).

LSOPT
Initialize only load step options.

ALL
Delete all loads and initialize all load step options and load factors.

Notes

Loads are deleted, and load step options are initialized to their default values.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All F.E. Loads
Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Inertia Lds
Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Loads & Opts
Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All SolidMod Lds
Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Reset Factors
Main Menu>Preprocessor>Loads>Load Step Opts>Reset Options
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>All
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Areas
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Elements
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Keypoints
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Lines
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Nodes

Main Menu>Solution>Define Loads>Delete>All Load Data>All F.E. Loads
Main Menu>Solution>Define Loads>Delete>All Load Data>All Inertia Lds
Main Menu>Solution>Define Loads>Delete>All Load Data>All Loads & Opts
Main Menu>Solution>Define Loads>Delete>All Load Data>All SolidMod Lds
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Reset Factors
Main Menu>Solution>Load Step Opts>Reset Options

LSDELE, *LSMIN*, *LSMAX*, *LSINC*

Deletes load step files.

SOLUTION: Load Step Operations

MP ME ST <> <> PR EM <> FL PP ED

LSMIN, *LSMAX*, *LSINC*
Range of load step files to be deleted, from *LSMIN* to *LSMAX* in steps of *LSINC*. *LSMAX* defaults to *LSMIN*, and *LSINC* defaults to 1. If *LSMIN* = ALL, all load step files are deleted (and *LSMAX* and *LSINC* are ignored). The load step files are assumed to be named **Jobname.S*n***, where *n* is a number assigned by the **LSWRITE** command (01--09,10,11, etc.). On systems with a 3-character limit on the extension, the "S" is dropped for numbers > 99.

Notes

Deletes load step files in the current directory (written by the **LSWRITE** command).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Delete LS Files
Main Menu>Solution>Define Loads>Operate>Delete LS Files

LSEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KSWP*

Selects a subset of lines.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

ALL
Restore the full set.

NONE
Unselect the full set.

INVE
Invert the current set (selected becomes unselected and vice versa).

STAT
Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Item
Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to LINE.

Comp
Component of the item (if required). Valid component labels are shown in the table below.

VMIN
Minimum value of item range. Ranges are line numbers, coordinate values, attribute numbers, etc., as appropriate for the item. If *VMIN* = 0.0, a tolerance of ±1.0E-6 is used, or ±0.005 x *VMIN* if *VMIN* = *VMAX*. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, ESYS, or NDIV and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **LLIST** command for a discussion of signed attributes.

VMAX
Maximum value of item range. *VMAX* defaults to *VMIN*. If *VMAX* ≠ *VMIN*, a tolerance of ±1.0E-8 x (*VMAX*-*VMIN*) is used.

VINC
Value increment within range. Used only with integer ranges (such as for line numbers). Defaults to 1. *VINC* cannot be negative.

KSWP
Specifies whether only lines are to be selected:

0
Select lines only.

1
Select lines, as well as keypoints, nodes, and elements associated with selected lines. Valid only with *Type* = S.

Command Default

All lines are selected.

Notes

Selects lines based on values of a labeled item and component. For example, to select a new set of lines based on line numbers 1 through 7, use **LSEL**,S,LINE,,1,7. The subset is used when the ALL label is entered (or implied)

on other commands, such as **LLIST**,ALL. Only data identified by line number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

If *Item* = LCCA, the command selects only those lines that were created by concatenation. The *KSWP* field is processed, but the *Comp*, *VMIN*, *VMAX*, and *VINC* fields are ignored.

If *Item* = HPT, the command selects only those lines that contain hard points.

Item = RADIUS is only valid for lines that are circular arcs.

LSEL is valid in any processor.

LSEL - Valid Item and Component Labels

Valid Item and Component Labels LSEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KSWP*

Item	Comp	Description
LINE		Line number.
EXT		Line numbers on exterior of selected area (ignore remaining fields).
LOC	X,Y,Z	X, Y, or Z center location in the active coordinate system.
TAN1	X,Y,Z	Unit vector component of outward tangent at beginning of line.
TAN2	X,Y,Z	Unit vector component of outward tangent at end of line.
NDIV		Number of divisions within the line.
SPACE		Spacing ratio of line divisions.
MAT		Material number associated with the line.
TYPE		Element type number associated with the line.
REAL		Real constant set number associated with the line.
ESYS		Element coordinate system associated with the line.
SEC		Cross section ID number. [**SECNUM**]
LENGTH		Length of the line.
RADIUS		Radius of the line.
HPT		Line number (selects only lines with associated hard points).
LCCA		Concatenated lines (selects only lines that were created by concatenation [**LCCAT**]).

Menu Paths

Utility Menu>Select>Entities

LSLA, *Type*

Selects those lines contained in the selected areas.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type

Label identifying the type of line select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

LSLK, *Type*, *LSKEY*

Selects those lines containing the selected keypoints.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of line select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

LSKEY
Specifies whether all contained line keypoints must be selected [**KSEL**]:

0
Select line if any of its keypoints are in the selected keypoint set.

1
Select line only if all of its keypoints are in the selected keypoint set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

LSOPER

Specifies "Load step operations" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Load Step Operations

/LSPEC, *LCOLOR*, *LINSTL*, *XLNWID*

Specifies annotation line attributes (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

LCOLOR

Line color:

0
: Black

1
: Red-Magenta

2
: Magenta

3
: Blue-Magenta

4
: Blue

5
: Cyan-Blue

6
: Cyan

7
Green-Cyan

8
Green

9
Yellow-Green

10
Yellow

11
Orange

12
Red

13
Dark Gray

14
Light Gray

15
White

LINSTL
Line style:

0
Solid line.

1
Dashed line.

XLNWID
Line width multiplier (1.0 to 20.0). Defaults to 1.0.

Notes

Specifies annotation line attributes to control certain characteristics of the lines created via the **/LINE**, **/LARC**, **/LSYMBOL**, **/POLYGON**, **/PMORE**, **/PCIRCLE** and **/PWEDGE** commands. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

LSREAD, *LSNUM*

Reads load and load step option data into the database.

SOLUTION: Load Step Operations

MP ME ST <> <> PR EM <> FL PP ED

LSNUM

Identification number of the load step file to be read. Defaults to 1 + highest number read in the current session. Issue **LSREAD**,STAT to list the current value of *LSNUM*. Issue **LSREAD**,INIT to reset *LSNUM* to 1. The load step files are assumed to be named **Jobname.S*n***, where *n* is a number assigned by the **LSWRITE** command (01--09,10,11, etc.). On systems with a 3-character limit on the extension, the "S" is dropped for *LSNUM* > 99.

Notes

Reads load and load step option data from the load step file into the database. **LSREAD** *will not* clear the database of *all* current loads. However, if a load is respecified with **LSREAD**, then it will overwrite the existing load. See the **LSWRITE** command to write load step files, and the **LSDELE** command to delete load step files. **LSREAD** removes any existing **SFGRAD** specification.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Read LS File
Main Menu>Solution>Load Step Opts>Read LS File

LSSCALE, *NL1*, *NL2*, *NINC*, *RX*, *RY*, *RZ*, *KINC*, *NOELEM*, *IMOVE*

Generates a scaled set of lines from a pattern of lines.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1, *NL2*, *NINC*

Set of lines (*NL1* to *NL2* in steps of *NINC*) that defines the pattern to be scaled. *NL2* defaults to *NL1*, *NINC* defaults to 1. If *NL1* = ALL, *NL2* and *NINC* are ignored and the pattern is defined by all selected lines. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

RX, *RY*, *RZ*

Scale factors to be applied to the X, Y, Z keypoint coordinates in active coordinate system (*RR*, *R*θ, *RZ* for cylindrical; *RR*, *R*θ, *R*Φ for spherical). Note that the *R*θ and *R*Φ scale factors are interpreted as angular *offsets*. For example, for **CSYS**,1, *RR*, *R*θ, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

KINC

Increment to be applied to keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned [**NUMSTR**].

NOELEM

Specifies whether nodes and elements are also to be generated:

0
Nodes and line elements associated with the original lines will be generated (scaled) if they exist.

1
Nodes and line elements will *not* be generated.

IMOVE
Specifies whether lines will be moved or newly defined:

0
Additional lines will be generated.

1
Original lines will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a scaled set of lines (and their corresponding keypoints and mesh) from a pattern of lines. The MAT, TYPE, REAL, and ESYS attributes are based on the lines in the pattern and not the current settings. Scaling is done in the active coordinate system. Lines in the pattern could have been generated in any coordinate system.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Scale>Lines

LSSOLVE, *LSMIN*, *LSMAX*, *LSINC*
Reads and solves multiple load steps.

SOLUTION: Load Step Operations

MP ME ST <> <> PR EM <> FL <> ED

LSMIN, *LSMAX*, *LSINC*
Range of load step files to be read and solved, from *LSMIN* to *LSMAX* in steps of *LSINC*. *LSMAX* defaults to *LSMIN*, and *LSINC* defaults to 1. If *LSMIN* is blank, a brief command description is displayed. The load step files are assumed to be named **Jobname.S*n***, where *n* is a number assigned by the **LSWRITE** command (01--09,10,11, etc.). On systems with a 3-character limit on the extension, the "S" is dropped for numbers > 99.

Notes

LSSOLVE invokes an ANSYS macro to read and solve multiple load steps. The macro loops through a series of load step files written by the **LSWRITE** command. The macro file called by **LSSOLVE** is called **LSSOLVE.MAC**.

LSSOLVE cannot be used with the birth-death option.

LSSOLVE does not support restarts.

LSSOLVE does not apply to FLOTRAN.

Menu Paths

Main Menu>Solution>Solve>From LS Files

LSTR, *P1*, *P2*

Defines a straight line irrespective of the active coordinate system.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

P1
Keypoint at the beginning of line. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI)

P2
Keypoint at the end of line.

Notes

Defines a straight line from *P1* to *P2* using the global Cartesian coordinate system. The active coordinate system will be ignored. The line shape is invariant with the coordinate system after it is generated. Lines may be redefined only if not yet attached to an area.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Straight Line

---

LSUM

Calculates and prints geometry statistics of the selected lines.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

Notes

Calculates and prints geometry statistics (length, centroid, moments of inertia, etc.) associated with the selected lines. Geometry items are reported in the global Cartesian coordinate system. A unit density is assumed, irrespective of any material associations [**LATT**, **MAT**]. Items calculated by **LSUM** and later retrieved by a ***GET** or ***VGET** command are valid only if the model is not modified after the **LSUM** command is issued.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Lines

---

LSWRITE, *LSNUM*

Writes load and load step option data to a file.

SOLUTION: Load Step Operations

MP ME ST <> <> PR EM <> FL PP ED

LSNUM

Number to be assigned to the load step file name for identification purposes. Defaults to 1 + highest *LSNUM* used in the current session. Issue **LSWRITE**,STAT to list the current value of *LSNUM*. Issue **LSWRITE**,INIT to reset to 1. The load step file will be named **Jobname.S*n***, where *n* is the specified *LSNUM* value (preceded by "0" for values 1-9). On systems with a 3-character limit on the file name extension, the "S" is dropped for *LSNUM* > 99.

Command Default

The default behavior of **LSWRITE** is dependent on the **SOLCONTROL** command. When solution control is on (default behavior), the **LSWRITE** command does NOT write the default values for commands affected by solution control. When solution control is off (**SOLCONTROL**,OFF), **LSWRITE** does write the default values for commands affected by solution control. See **SOLCONTROL** for a list of those commands.

Notes

Writes all load and load step option data for the selected model to a load step file for later use. **LSWRITE** does *not* capture changes made to real constants (**R**) or material properties (**MP**). Solid model loads will not be saved if the model is not meshed. Solid model loads, if any, are transferred to the finite element model. Issue **LSCLEAR**,FE to delete finite element loads. One file is written for each load step. Use the **LSREAD** command to read a single load step file, and the **LSDELE** command to delete load step files. Use the **LSSOLVE** command to read and solve the load steps sequentially.

LSWRITE cannot be used with the birth-death option.

LSWRITE does not apply to FLOTRAN.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Write LS File
Main Menu>Solution>Load Step Opts>Write LS File

/LSYMBOL, *X*, *Y*, *SYMANG*, *SYMTYP*, *SYMSIZ*, *KEYBMP*

Creates annotation symbols (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

X

X location for symbol (-1.0 < X < 2.0).

Y

Y location for symbol (-1.0 < Y < 1.0).

SYMANG
Symbol orientation angle.

SYMTYP
Symbol type:

1
Arrow.

2
Tee.

3
Circle.

4
Triangle.

5
Star.

SYMSIZ
Symbol size multiplier (0.1 to 20.0). Defaults to 1.0.

KEYBMP
If *KEYBMP* = 1, the annotation is a bitmap. *SYMTYP* will then be a number from 1-99, indicating the bitmap type (see notes), and *X* and *Y* will define the lower left corner of the bitmap. The *SYMANG*, *SYMSIZ*arguments are ignored. If *KEYBMP* = 0, or blank, then the argument definitions above apply.

Notes

Defines annotation symbols to be written directly onto the display at a specified location. This is a command generated by the GUI and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All symbols are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** command to set the attributes of the symbol.

The *KEYBMP* argument reads the symtype argument to determine which bitmap to insert. This bitmap is defined by an integer between 1 and 99. Numbers 1 through 40 correspond to the standard texture values found in the **/TXTRE** command, while numbers 51 through 99 correspond to user supplied bitmaps, as defined using the Filename option of the **/TXTRE** command. Numbers 51 through 57 are predefined (the logos, clamps and arrows available from the GUI) but can be overridden. Numbers 41 through 50 are reserved.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

LSYMM, *Ncomp*, *NL1*, *NL2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Generates lines from a line pattern by symmetry reflection.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

Ncomp

Symmetry key:

X
: X symmetry (default).

Y
: Y symmetry.

Z
: Z symmetry.

NL1, *NL2*, *NINC*

Reflect lines from pattern beginning with *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and pattern is all selected lines [**LSEL**]. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

KINC

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM

Specifies whether nodes and elements are also to be generated:

0
: Generate nodes and elements associated with the original lines, if they exist.

1
: Do not generate nodes and elements.

IMOVE

Specifies whether areas will be moved or newly defined:

0
: Generate additional lines.

1
: Move original lines to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a reflected set of lines (and their corresponding keypoints and mesh) from a given line pattern by a symmetry reflection (see analogous node symmetry command, **NSYM**). The MAT, TYPE, REAL, and ESYS attributes are based upon the lines in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. The active coordinate system must be Cartesian. Lines in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Lines are generated as described in the **LGEN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Lines

LTAN, *NL1*, *P3*, *XV3*, *YV3*, *ZV3*

Generates a line at the end of, and tangent to, an existing line.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

NL1

Number of the line the generated line is tangent to. If negative, assume *P1* (see below), instead of *P2*, is the second keypoint of line *NL1*. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P3

Keypoint at which generated line must end.

The following fields are used only if a specified slope on the end of the new line is desired, otherwise a zero curvature end slope will be automatically calculated. To specify the end slope, use the following fields to define a "slope vector" that has its tail at the origin and its head at the point *XV*,*YV*,*ZV* in the active coordinate system [**CSYS**]. The corresponding end slope of the line will then be parallel to this "slope vector."

XV3, *YV3*, *ZV3*

Location (in the active coordinate system) of the head of the "slope vector" corresponding to the slope at the *P3* end of the line. The tail of the vector is at the coordinate system origin.

Notes

Generates a line (*P2-P3*) tangent at end point (*P2*) of line *NL1* (*P1-P2*).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Tangent to Line

---

LTRAN, *KCNTO*, *NL1*, *NL2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Transfers a pattern of lines to another coordinate system.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

KCNTO

Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system. The coordinate system type and parameters of *KCNTO* must be the same as the active system.

NL1, *NL2*, *NINC*

Transfer lines from pattern beginning with *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and pattern is all selected lines [**LSEL**]. If *NL1* = P, graphical picking is

enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

KINC
Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM
Specifies whether nodes and elements are also to be generated:

0
Generate nodes and elements associated with the original lines, if they exist.

1
Do not generate nodes and elements.

IMOVE
Specifies whether lines will be moved or newly defined:

0
Generate additional lines.

1
Move original lines to new position retaining the same keypoint numbers (*KINC* and *NOELM* are ignored). Valid only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Transfers a pattern of lines (and their corresponding keypoints and mesh) from one coordinate system to another (see analogous node transfer command, **TRANSFER**). The MAT, TYPE, REAL, and ESYS attributes are based upon the lines in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Coordinate and slope values are interpreted in the active coordinate system and are transferred directly. Lines are generated as described in the **LGEN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Lines

---

LUMPM, *Key*
Specifies a lumped mass matrix formulation.

SOLUTION: Analysis Options

MP ME ST <> <> PR <> <> <> PP ED

Key
Formulation key:

OFF
Use the element-dependent default mass matrix formulation (default).

ON
Use a lumped mass approximation.

Command Default

Use the default element mass matrix.

Notes

This command is also valid in PREP7. If used in SOLUTION, this command is valid only within the first load step.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Preprocessor>Loads>Analysis Type>New Analysis
Main Menu>Solution>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>New Analysis

LVSCALE, *FACT*

Scales the load vector for mode superposition analyses.

SOLUTION: Dynamic Options

MP ME ST <> <> PR <> <> <> PP ED

FACT
Scale factor applied to load vector. Defaults to 0.0.

Command Default

No load vector is applied.

Notes

Specifies the scale factor for the load vector that was created in a modal (**ANTYPE**,MODAL) analysis. Applies only to the mode superposition transient (**ANTYPE**,TRANS), mode superposition harmonic response (**ANTYPE**,HARMIC), and PSD spectrum (pressure PSD) analyses.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Load Vector>For Mode Super
Main Menu>Preprocessor>Loads>Define Loads>Apply>Load Vector>For PSD
Main Menu>Preprocessor>Loads>Define Loads>Delete>Load Vector>For Mode Super
Main Menu>Preprocessor>Loads>Define Loads>Delete>Load Vector>For PSD
Main Menu>Solution>Define Loads>Apply>Load Vector>For Mode Super
Main Menu>Solution>Define Loads>Apply>Load Vector>For PSD
Main Menu>Solution>Define Loads>Delete>Load Vector>For Mode Super
Main Menu>Solution>Define Loads>Delete>Load Vector>For PSD

LWPLAN, *WN*, *NL1*, *RATIO*

Defines the working plane normal to a location on a line.

DATABASE: Working Plane

MP ME ST DY <> PR EM <> FL PP ED

WN

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified.

NL1

Number of line to be used. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

RATIO

Location on *NL1*, specified as a ratio of the line length. Must be between 0.0 and 1.0. If *RATIO* = P, use graphical picking to specify location on the line.

Command Default

Working plane is parallel to the Global X-Y plane at Z = 0.0.

Notes

Defines a working plane (to assist in picking operations) normal to a location on a line. See **WPSTYL** command to set the style of working plane display.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Align WP with>Plane Normal to Line

M Commands

M, *NODE*, *Lab1*, *NEND*, *NINC*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

Defines master degrees of freedom for reduced and superelement generation analyses.

SOLUTION: Master DOF

MP ME ST <> <> PR <> <> <> PP ED

NODE

Node number at which master degree of freedom is defined. If ALL, define master degrees of freedom at all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab1

Valid degree of freedom label. If ALL, use all appropriate labels. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Electric labels: VOLT (voltage).

NEND, *NINC*

Define all nodes from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1) as master degrees of freedom in the specified direction.

Lab2, *Lab3*, *Lab4*, *Lab5*, *Lab6*

Additional master degree of freedom labels. The nodes defined are associated with each label specified.

Notes

Defines master degrees of freedom (MDOF) for reduced (dynamic and superelement generation) analyses. If defined for other analyses, MDOF are ignored. If used in SOLUTION, this command is valid only within the first load step. Scalar MDOF (VOLT) are not permitted in structural analyses if mass or damping matrices are reduced. Reduced analyses normally apply only to the UX, UY, UZ, ROTX, ROTY, and ROTZ labels.

Repeat **M** command for additional master degrees of freedom. Limit is equal to the maximum in-memory wavefront size (see the *ANSYS Basic Analysis Guide*).

The reduced modal (**ANTYPE**,MODAL), reduced transient (**ANTYPE**,TRANS), reduced harmonic response (**ANTYPE**,HARMIC), and the substructure (**ANTYPE**,SUBSTR) analyses utilize the matrix condensation technique to reduce the structure matrices to those characterized by a set of master degrees of freedom.

Master degrees of freedom are identified by a list of nodes and their nodal directions. The actual degree of freedom directions available for a particular node depends upon the degrees of freedom associated with element types [**ET**] at that node. For example, degrees of freedom available with BEAM3 elements are UX, UY, and ROTZ only. There must be some mass (or stress stiffening in the case of the buckling analysis) associated with each master degree of freedom (except for the VOLT label). The mass may be due either to the distributed mass of the element or due to discrete lumped masses at the node. If a master degree of freedom is specified at a constrained point, it is ignored. If a master degree of freedom is specified at a coupled node, it should be specified at the prime node of the coupled set. Master degrees of freedom can also be generated automatically (during solution) by issuing the **TOTAL** command in PREP7 or SOLUTION.

Transient displacements and forces, used to apply motion to a structure in the reduced transient or reduced harmonic response analysis, must be applied at a master degree of freedom. Substructure analysis connection points must be defined as master degrees of freedom.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Master DOFs>User Selected>Define
Main Menu>Preprocessor>Modeling>CMS>CMS Interface>Define
Main Menu>Solution>Master DOFs>User Selected>Define

MAGOPT, *VALUE*

Specifies options for a 3-D magnetostatic field analysis.

SOLUTION: Load Step Options

MP ME ST <> <> <> EM <> <> PP ED

VALUE
Option key:

0
Calculate a complete H field solution in the entire domain using a single (reduced) potential.

Caution: When used in problems with both current sources and iron regions, errors may result due to numerical cancellation.

1
Calculate and store a preliminary H field in "iron" regions ($\mu_r \neq 1$). Requires flux-parallel boundary conditions to be specified on exterior iron boundaries. Used in conjunction with subsequent solutions with *VALUE* = 2 followed by *VALUE* = 3. Applicable to multiply-connected iron domain problems.

2
Calculate and store a preliminary H field in "air" regions ($\mu_r = 1$). The air-iron interface is appropriately treated internally by the program. Used in conjunction with a subsequent solution with *VALUE* = 3. Applicable to singly-connected iron domain problems (with subsequent solution with *VALUE* = 3) or to multiply-connected iron domain problems (when preceded by a solution with *VALUE* = 1 and followed by a solution with *VALUE* = 3).

3
Use the previously stored H field solution(s) and calculate the complete H field.

Notes

Specifies the solution sequence options for a 3-D magnetostatic field analysis using a scalar potential (MAG). The solution sequence is determined by the nature of the problem.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>DSP Method
Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>GSP Method
Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>RSP Method
Main Menu>Solution>Load Step Opts>Magnetics>Options Only>DSP Method

Main Menu>Solution>Load Step Opts>Magnetics>Options Only>GSP Method
Main Menu>Solution>Load Step Opts>Magnetics>Options Only>RSP Method

MAGSOLV, *OPT*, *NRAMP*, *CNVCSG*, *CNVFLUX*, *NEQIT*, *BIOT*,*CNVTOL*

Specifies magnetic solution options and initiates the solution.

SOLUTION: Load Step Options

MP ME ST <> <> <> EM <> <> PP ED

OPT

Static magnetic solution option:

0

Vector potential (MVP) or edge formulation (default).

1

Combined vector potential and reduced scalar potential (MVP-RSP).

2

Reduced scalar potential (RSP).

3

Difference scalar potential (DSP).

4

General scalar potential (GSP).

NRAMP

Number of ramped substeps for the first load step of a nonlinear MVP or MVP-RSP solution. Defaults to 3. If *NRAMP* = -1, ignore the ramped load step entirely.*NRAMP* is ignored for linear magnetostatics.

CNVCSG

Tolerance value on the program-calculated reference value for the magnetic current-segment convergence. Used for the MVP, the MVP-RSP, and the edge formulation solution options (*OPT* = 0 and 1). Defaults to 0.001.

CNVFLUX

Tolerance value on the program-calculated reference value for the magnetic flux convergence. Used for all scalar potential solution options (*OPT* = 2, 3, 4). Defaults to 0.001.

NEQIT

Maximum number of equilibrium iterations per load step. Defaults to 25.

BIOT

Option to force execution of a Biot-Savart integral solution [**BIOT**,NEW] for the scalar potential options. Required if multiple load steps are being performed with different current source primitives (SOURC36 elements).

0

Do not force execution of Biot-Savart calculation (default); Biot-Savart is automatically calculated only for the first solution.

1

Force execution of Biot-Savart calculation.

CNVTOL

Sets the convergence tolerance for AMPS reaction. Defaults to 1e-3. Applicable only when KEYOPT(1) = 5 on SOLID117.

Notes

MAGSOLV invokes an ANSYS macro which specifies magnetic solution options and initiates the solution. The macro is applicable to any ANSYS magnetostatic analysis using the magnetic vector potential (MVP), reduced scalar potential (RSP), difference scalar potential (DSP), general scalar potential (GSP), or combined MVP-RSP formulation options. Results are only stored for the final converged solution. (In POST1, issue ***SET**,LIST to identify the load step of solution results.) The macro internally determines if a nonlinear analysis is required based on magnetic material properties.

If you use the BIOT option and issue **SAVE** after solution or postprocessing, the Biot-Savart calculations are saved to the database, but will be overwritten upon normal exit from the program. To save this data after issuing **SAVE**, use the **/EXIT**,NOSAVE command. You can also issue the **/EXIT**,SOLU command to exit ANSYS and save all solution data, including the Biot-Savart calculations, in the database. Otherwise, when you issue **RESUME**, the Biot-Savart calculation will be lost (resulting in a zero solution).

The MVP, MVP-RSP, and edge formulation options perform a two-load-step solution sequence. The first load step ramps the applied loads over a prescribed number of substeps (*NRAMP*), and the second load step calculates the converged solution. For linear problems, only a single load step solution is performed. The ramped load step can be bypassed by setting *NRAMP* to -1.

The RSP option solves in a single load step using the adaptive descent procedure. The DSP option uses two load steps, and the RSP solution uses three load steps.

The following analysis options and nonlinear options are controlled by this macro: **KBC**, **NEQIT**, **NSUBST**, **CNVTOL**, **NROPT**, **MAGOPT**, and **OUTRES**.

Menu Paths

Main Menu>Solution>Solve>Electromagnet>Static Analysis>Opt&Solv

---

MASTER

Specifies "Master DOF" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Master DOF

MAT, *MAT*

Sets the element material attribute pointer.

PREP7: Meshing
PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

MAT
Assign this material number to subsequently defined elements (defaults to 1).

Command Default

MAT = 1.

Notes

Identifies the material number to be assigned to subsequently defined elements. This number refers to the material number (*MAT*) defined with the material properties [**MP**]. Material numbers may be displayed [**/PNUM**].

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs
Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

---

MATER

Specifies "Material properties" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Materials

MCHECK, *Lab*

Checks mesh connectivity.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

Lab

Operation:

ESEL

Unselects the valid elements.

Notes

Wherever two area or volume elements share a common face, **MCHECK** verifies that the way the elements are connected to the face is consistent with their relative normals or integrated volumes. (This may detect folds or otherwise overlapping elements.)

MCHECK verifies that the element exterior faces form simply-connected closed surfaces. (This may detect unintended cracks in a mesh.)

MCHECK warns if the number of element facets in a 2-D loop or 3-D shell is not greater than a computed limit. This limit is the smaller of either three times the number of faces on one element, or one-tenth the total number of element faces in the model. (This may detect holes in the middle of a mesh.)

The **MCHECK** command will perform a number of validity checks on the selected elements, including

1. Normal check: Wherever two area elements share a common edge, **MCHECK** verifies that the ordering of the nodes on each element is consistent with their relative normals.
2. Volume check: Wherever two volume elements share a common face, **MCHECK** verifies that the sign of the integrated volume of each element is consistent.
3. Closed surface check: **MCHECK** verifies that the element exterior faces form simply-connected closed surfaces (this may detect unintended cracks in a mesh).
4. Check for holes in the mesh: **MCHECK** warns if the number of element faces surrounding an interior void in the mesh is small enough to suggest one or more accidentally omitted elements, rather than a deliberately formed hole. For this test, the number of faces around the void is compared to the smaller of a) three times the number of faces on one element, or b) one-tenth the total number of element faces in the model.

Menu Paths

Main Menu>Preprocessor>Meshing>Check Mesh>Connectivity>Ck Connectvty
Main Menu>Preprocessor>Meshing>Check Mesh>Connectivity>Sel Bad Connt

MDAMP, *STLOC*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

Defines the damping ratios as a function of mode.

SOLUTION: Dynamic Options

MP ME ST <> <> PR <> <> <> PP ED

STLOC
Starting location in table for entering data. For example, if *STLOC* = 1, data input in the *V1* field applies to the first constant in the table. If *STLOC* = 7, data input in the *V1* field applies to the seventh constant in the table, etc. Defaults to the last location filled + 1.

V1, *V2*, *V3*, *V4*, *V5*, *V6*
Data assigned to six locations starting with *STLOC*. If a value is already in this location, it will be redefined. Blank values for *V2* to *V6* leave the corresponding previous value unchanged.

Notes

Defines the damping ratios as a function of mode. Table position corresponds to mode number. These ratios are added to the **DMPRAT** value, if defined. Use **STAT** command to list current values. Applies to the mode superposition harmonic response (**ANTYPE**,HARMIC), the mode superposition linear transient dynamic (**ANTYPE**,TRANS), and the spectrum (**ANTYPE**,SPECTR) analyses. Repeat **MDAMP** command for additional constants (300 maximum).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping
Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping

MDELE, *NODE*, *Lab1*, *NEND*, *NINC*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

Deletes master degrees of freedom.

SOLUTION: Master DOF

MP ME ST <> <> PR <> <> <> PP ED

NODE, *Lab1*, *NEND*, *NINC*
Delete master degrees of freedom in the *Lab1* direction [**M**] from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1). If *NODE* = ALL, *NEND* and *NINC* are ignored and masters for all selected nodes [**NSEL**] are deleted. If *Lab1* = ALL, all label directions will be deleted. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab2, *Lab3*, *Lab4*, *Lab5*, *Lab6*
Delete masters in these additional directions.

Notes

Deletes master degrees of freedom. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Master DOFs>User Selected>Delete
Main Menu>Solution>Master DOFs>User Selected>Delete

MDPLOT, *Function*, *Dmpname*, *Scale*

Plots frequency-dependent modal damping coefficients calculated by DMPEXT.

SOLUTION: Dynamic Options

MP ME <> <> <> <> <> <> <> PP ED

Function
Function to display.

d_coeff
Damping coefficient

s_coeff
Squeeze coefficient

d_ratio
Damping ratio

s_ratio
Squeeze stiffness ratio

Dmpname
Array parameter name where damping information is stored. Defaults to **d_damp**.

Scale
Indicates whether to perform a linear or a double logarithmic plot.

LIN
Perform a linear plot. Default

LOG
Perform a double logarithmic plot.

Notes

See Chapter 16, " Thin Film Analysis" for more information on thin film analyses.

Menu Paths

Main Menu>General Postproc>Plot Results>ThinFilm>Plot Dmp Parm

---

MEMM, *Lab*, *Kywrd*

Allows the current session to keep allocated memory

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Lab

When *Lab* = KEEP, the memory manager's ability to acquire and keep memory is controlled by *Kywrd*

Kywrd

Turns the memory "keep" mode on or off

ON

Keep any memory allocated during the analysis.

OFF

Use memory dynamically and free it up to other users after use (default).

Command Default

Kywrd = OFF.

Notes

You can use the **MEMM** command to ensure that memory intensive operations will always have the same memory available when the operations occur intermittently. Normally, if a large amount of memory is allocated for a specific operation, it will be returned to the system once the operation is finished. This option always maintains the highest level used during the analysis until the analysis is finished.

The **MEMM** command does not affect the value you specify with the `-m` switch. When you allocate memory with the `-m` switch, that amount will always be available. However, if dynamic memory allocation in excess of the `-m` value occurs, you can use the **MEMM** command to ensure that amount is retained until the end of your analysis.

Menu Paths

This command cannot be accessed from a menu.

/MENU, *Key*

Activates the Graphical User Interface (GUI).

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Key

Activation key:

ON

Activates the menu system (device dependent).

GRPH

Enters non-GUI graphics mode. This option is intended for use on graphics devices that do not support the full Motif-based GUI.

Command Default

GUI is on if entering the ANSYS program through the launcher. GUI is off if entering using the ANSYS execution command.

Notes

Activates the Graphical User Interface (GUI).

> **Caution:** if you include the **/MENU**,ON command in your start81.ans, it should be the *last* command in the file. Any commands after **/MENU**,ON may be ignored. (It is not necessary to include the **/SHOW** and **/MENU**,ON commands in start81.ans if you will be using the launcher to enter the ANSYS program.)

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

MESHING
Specifies "Meshing" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Meshing

MFANALYSIS, *Key*
Turns a multifield analysis on or off.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

Key
Multifield analysis key:

ON
Activates a multifield analysis.

OFF
Deactivates a multifield analysis (default).

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Setup>Global
Main Menu>Solution>MultiField Set Up>Setup>Global

MFBUCKET, *Key*, *Value*

Turns a bucket search on or off.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

Key
Bucket search key:

ON
Activates a bucket search (default).

OFF
Deactivates a bucket search. A global search is then activated.

Value
Scaling factor (%) used to determine the number of buckets for a bucket search. Defaults to 50%.

Notes

A bucket search will more efficiently compute the mapping of surface and volumetric interpolation data across field interfaces (flagged by the FSIN label using **SF**, **SFA**, **SFE**, or **SFL** or the FVIN label using **BFE**).

The number of buckets used to partition a flagged interface is equal to the scaling factor (%) times the total number of interface elements. For example, for the default scaling factor of 50% and a 10,000 element interface, 5,000 buckets are used.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Setup>Global
Main Menu>Solution>MultiField Set Up>Setup>Global

MFCALC, *FNUMB*, *FREQ*

Specifies a calculation frequency for a field in a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

FNUMB

Field number set by the **MFELEM** command.

FREQ

Perform calculation every Nth multifield time step. Defaults to 1 for every time step.

Notes

This command only applies to a harmonic analysis of the specified field. It is useful when a field contributes negligible field interaction within a single multifield time step.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Frequency
Main Menu>Solution>MultiField Set Up>Frequency

MFCLEAR, *Option*, *Value*

Deletes multifield analysis settings.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

Option

SOLU
Resets all ANSYS solution commands to their default states. This clears analysis options when setting up different fields for a multifield analysis.

FIELD
Deletes all multifield specifications for the specified field number.

SINT
Deletes all multifield specifications for the specified surface interface number.

VINT
Deletes all multifield specifications for the volumetric interface number.

ORD
Deletes the analysis order specified by the **MFORDER** command.

EXT
Deletes external fields specified by the **MFEXTER** command

Value

Use only for *Option* = FIELD, SINT, or VINT.

Option = FIELD deletes all multifield specifications for the field number *Value*. *Value* defaults to 0 (no deletions). A *Value* of -1 deletes the multifield settings for all fields.

Option = SINT deletes surface interface specifications for the surface interface number *Value*. *Value* defaults to 0 (no deletions). A *Value* of -1 deletes all surface interface specifications. This command deletes the multifield commands, not the boundary conditions themselves.

Option = VINT deletes all volumetric interface specifications for the volume interface number *Value*. *Value* defaults to 0 (no deletions). A *Value* of -1 deletes all volumetric interface specifications. This command deletes the multifield commands, not the boundary conditions themselves.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Clear
Main Menu>Solution>MultiField Set Up>Clear

MFCMMAND, *FNUMB*, *Fname*, *Ext*

Captures field solution options in a command file.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

FNUMB
Field number specified by the **MFELEM** command.

Fname
Command file name specified for the field number. Defaults to field"FNUMB".

Ext
Extension for *Fname*. Defaults to .cmd.

Notes

All relevant solution option commands for the specified field are written to a file with the extension .cmd. Refer to the commands in the following tables in the *ANSYS Commands Reference*: Analysis Options, Nonlinear Options, Dynamic Options, and Load Step Options.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Capture
Main Menu>Solution>MultiField Set Up>Capture

MFCONV, *Lab*, *VALUE*

Sets convergence values for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

Lab

Valid labels:

Force labels: FX, FY, FZ
Displacement labels: UX, UY, UZ
Temperature label: TEMP
Heat flux label: HFLU (for surface load transfer)
Heat generation label: HGEN (for volume load transfer)

If *Lab* = ALL, **MFCONV** applies the convergence value to all nine variables.

VALUE

Convergence value. Defaults to 0.001.

Notes

MFCONV sets convergence values for variables at the multifield interface.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Stagger>Convergence
Main Menu>Solution>MultiField Set Up>Stagger>Convergence

MFDTIME, *DTIME*

Sets time step increment for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

DTIME

Time step increment. Defaults to 1.

Notes

This command specifies time increments for a multifield analysis. The time step size remains constant for a multifield analysis. You can use a smaller time step within each field analysis. This is called subcycling. Use the **DELTIM** and **AUTOTS** commands to subcycle a structural, thermal, or electromagnetic analysis. Use the **FLDATA4** command to subcycle a fluid analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Time Ctrl
Main Menu>Solution>MultiField Set Up>Time Ctrl

MFELEM, *FNUMB*, *ITYPE1*, *ITYPE2*, *ITYPE3*, *ITYPE4*, *ITYPE5*, *ITYPE6*, *ITYPE7*, *ITYPE8*, *ITYPE9*, *ITYPE10*
Defines a field by grouping element types.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

FNUMB
Field number for a group of element types.

ITYPE1, *ITYPE2*, *ITYPE3*, *ITYPE4*, *ITYPE5*, *ITYPE6*, *ITYPE7*, *ITYPE8*, *ITYPE9*, *ITYPE10*
Element types defined by the **ET** command.

Notes

You can define up to ten element types per field.

Define only element types that contain elements in the field. Do not include MESH200 because it is a "mesh-only" element that does not contribute to the solution.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Define
Main Menu>Solution>MultiField Set Up>Define

MFEXTER, *FNUMB1*, *FNUMB2*, *FNUMB3*, *FNUMB4*, *FNUMB5*, *FNUMB6*, *FNUMB7*, *FNUMB8*, *FNUMB9*, *FNUMB10*, *FNUMB11*, *FNUMB12*, *FNUMB13*, *FNUMB14*, *FNUMB15*, *FNUMB16*, *FNUMB17*, *FNUMB18*, *FNUMB19*, *FNUMB20*
Defines external fields for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

FNUMB1, *FNUMB2*, *FNUMB3*, *FNUMB4*, *FNUMB5*, *FNUMB6*, *FNUMB7*, *FNUMB8*, *FNUMB9*, *FNUMB10*, *FNUMB11*, *FNUMB12*, *FNUMB13*, *FNUMB14*, *FNUMB15*, *FNUMB16*, *FNUMB17*, *FNUMB18*, *FNUMB19*, *FNUMB20*
External field numbers defined by the **MFELEM** command.

Notes

This command specifies external field numbers to be used for load transfer in a multifield analysis. Use the **MFIMPORT** command to import the external fields.

Use the **MFELEM** command to specify external field numbers. Use the **MFORDER** command to specify the solution order for the external fields.

You can define a maximum of 20 fields.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Setup>External
Main Menu>Solution>MultiField Set Up>Setup>External

MFFNAME, *FNUMB*, *Fname*

Specifies a file name for a field in a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

FNUMB
Field number specified by the **MFELEM** command.

Fname
File name. Defaults to field"FNUMB".

Notes

All files created for the field will have this file name with the appropriate extensions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Define
Main Menu>Solution>MultiField Set Up>Define

MFIMPORT, *FNUMB*, *Option*, *Fname*, *Ext*

Imports a new field into a current multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

FNUMB
Field number specified by the **MFELEM** command.

Option
Selects data to read.

DB
Reads a CDB file. The CDB file name and extension are specified by *Fname* and *Ext*.

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to CDB if *Fname* is blank.

Notes

The field to be imported should be written to a CDB file (**CDWRITE** command). This file is read into the database, offsetting all existing element type numbers, node numbers, etc. in order to accommodate the imported field. (See the **NUMOFF** command for information on offset capabilities.) It then updates all of the previously issued MFxx commands to the new element type numbers. A new field is created using the specified field number, which must not currently exist. If there are no multifield command files written for the existing fields in the database, one will be written for each field with the default name (see the **MFCMMAND** command). A **MFCMMAND** will be issued for the imported field as well.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Import
Main Menu>Solution>MultiField Set Up>Import

MFINTER, *Option*

Specifies the interface load transfer interpolation option for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

Option
Interface load transfer option:

CONS
Conservative formulation for load transfer.

NONC
Nonconservative formulation for load transfer (default).

Notes

For more information on conservative and nonconservative load transfer, see Load Transfer in the *ANSYS Coupled-Field Analysis Guide*.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Setup>Global
Main Menu>Solution>MultiField Set Up>Setup>Global

MFITER, *ITER*

Sets the maximum number of stagger iterations for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

ITER

Maximum number of iterations. Defaults to 10.

Notes

The maximum number of stagger iterations applies to each time step in a multifield analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Stagger>Max Iterations
Main Menu>Solution>MultiField Set Up>Stagger>Max Iterations

MFLIST, *Option*, *Value*

Lists the settings for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

Option

ALL
Lists all multifield analysis options.

SOLU
Lists all solution related multifield options.

FIELD
Lists all multifield options related to the specified field number.

SINT
Lists all surface interface information for the specified surface interface number.

VINT
Lists all volumetric interface information for the specified volumetric interface number.

Value

Use only for *Option* = FIELD, SINT, or VINT.

Option = FIELD lists all multifield options for the field number *Value*. Value defaults to 0, which lists information for all fields.

Option = SINT lists all surface interface information for the interface number *Value*. Value defaults to 0, which lists information for all surface interfaces.

Option = VINT lists all volumetric interface information for interface number *Value*. Value defaults to 0, which lists information for all volumetric interfaces.

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Status
Main Menu>Solution>MultiField Set Up>Status

MFMAP, *Lab1*, *Lab2*, *Filename*, *Opt*

Calculates, saves, resumes, or deletes mapping data in a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

Lab1
Operation label:

CALC
Calculate mapping data and keep it in memory (default).

SAVE
Calculate mapping data, keep it in memory, and save it to a file. (If MFMAP,CALC or MFMAP,RESU have been issued, just save it to a file.)

RESU
Resume the mapping from a file and keep it in memory.

DELE
Free the mapping memory.

Lab2
Applicable mapping label:

ALL
Surface and volumetric mapping.

SURF
Surface mapping only.

VOLU
Volumetric mapping only.

Filename
The file name for a mapping data file (**filename.sur** for surface mapping and **filename.vol** for volumetric mapping). Defaults to Jobname. Applies to the commands MFMAP,SAVE and MFMAP,RESU only.

Opt
File format:

BINA
Binary file (default).

ASCI
ASCII file.

Notes

This command calculates, saves, resumes, or deletes mapping data. It defaults to calculating the mapping data. If **MFMAP** has not been previously issued, the mapping data will be automatically calculated during the solution process. On the other hand, the multifield solver will use previously created mapping data. Resumed mapping files must have load transfer specifications that are consistent with those of the current **MFSURFACE** and **MFVOLUME** commands and the ANSYS database.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Interface>Mapping
Main Menu>Solution>MultiField Set Up>Interface>Mapping

MFORDER, *FNUMB1*, *FNUMB2*, *FNUMB3*, *FNUMB4*, *FNUMB5*, *FNUMB6*, *FNUMB7*, *FNUMB8*, *FNUMB9*, *FNUMB10*, *FNUMB11*, *FNUMB12*, *FNUMB13*, *FNUMB14*, *FNUMB15*, *FNUMB16*, *FNUMB17*, *FNUMB18*, *FNUMB19*, *FNUMB20*
Specifies field solution order for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

FNUMB1, *FNUMB2*, *FNUMB3*, *FNUMB4*, *FNUMB5*, *FNUMB6*, *FNUMB7*, *FNUMB8*, *FNUMB9*, *FNUMB10*, *FNUMB11*, *FNUMB12*, *FNUMB13*, *FNUMB14*, *FNUMB15*, *FNUMB16*, *FNUMB17*, *FNUMB18*, *FNUMB19*, *FNUMB20*
Field numbers defined by the **MFELEM** command .

Notes

You can define up to twenty fields in a multifield analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Setup>Order
Main Menu>Solution>MultiField Set Up>Setup>Order

MFOUTPUT, *FREQ*
Specifies results file output frequency for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

FREQ

N
Write solution every Nth (and the last) time step. Defaults to 1, for every time step.

Notes

A **MFOUTPUT** setting overrides any other output frequency setting (**OUTRES**). To select the solution items, use the **OUTRES** command.

FLOTRAN ignores a **MFOUTPUT** setting. You should write out every time step in a FLOTRAN analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Frequency
Main Menu>Solution>MultiField Set Up>Frequency

MFRELAX, *Lab*, *VALUE*

Sets relaxation values for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

Lab
Valid labels:

Force label: FORC
Displacement label: DISP
Temperature label: TEMP
Heat flux label: HFLU
Heat generation label: HGEN

If *Lab* = ALL, **MFRELAX** applies the relaxation value to all variables.

VALUE
Relaxation value. Defaults to 0.5 for all labels.

Notes

MFRELAX sets relaxation values for the load transfer variables at a surface or volume interface.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Stagger>Relaxation
Main Menu>Solution>MultiField Set Up>Stagger>Relaxation

MFRSTART, *TIME*

Specifies a restart time for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

TIME
Restart time. Defaults to 0 (new analysis).

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Time Ctrl
Main Menu>Solution>MultiField Set Up>Time Ctrl

MFSURFACE, *INUMB*, *FNUMB1*, *Label*, *FNUMB2*

Defines a surface load transfer for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

INUMB
Interface number for load transfer. The interface number corresponds to the interface number specified by the surface flag FSIN (SFxxcommands).

FNUMB1
Field number of sending field.

Label
Valid surface load labels:

Force label: FORC
Displacement label: DISP
Temperature label: TEMP
Heat flux label: HFLU

FNUMB2
Field number for receiving field.

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Interface>Surface
Main Menu>Solution>MultiField Set Up>Interface>Surface

MFTIME, *TIME*

Sets end time for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

TIME
End time of a multifield analysis. Defaults to 1.

Notes

A **MFTIME** setting overrides any other end time setting (**TIME** or **FLDATA4**).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Time Ctrl
Main Menu>Solution>MultiField Set Up>Time Ctrl

MFTOL, *Key*, *Value*

Turns normal distance checking on for surface mapping in a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

Key
Normal distance key

ON
Activates normal distance checking.

OFF
Deactivates normal distance checking (default).

Value
The normal distance tolerance for surface or volumetric mapping. Defaults to 1.0e-6.

Notes

For a dissimilar mesh interface, the nodes of one mesh are mapped to the local coordinates of an element in the other mesh. When normal distance checking is activated, the mapping tool checks the normal distance from the node to the nearest element. The node is considered improperly mapped if the normal distance exceeds the tolerance value. The mapping tool creates a component to graphically display the improperly mapped nodes. See Section 4.2.2.2: Mapping Diagnostics in the *ANSYS Coupled-Field Analysis Guide* for more information.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Setup>Global

Main Menu>Solution>MultiField Set Up>Setup>Global

MFVOLUME, *INUMB*, *FNUMB1*, *Label*, *FNUMB2*

Defines a volume load transfer for a multifield analysis.

SOLUTION: Multifield

MP <> <> <> <> <> <> <> <> PP ED

INUMB
Interface number for load transfer. The interface number corresponds to the interface number specified by the volume flag FVIN (**BFE** command).

FNUMB1
Field number of sending field.

Label
Valid volume load labels:

Force label: FORC
Displacement label: DISP
Temperature label: TEMP
Heat generation: HGEN

FNUMB2
Field number for receiving field.

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>MultiField Set Up>Interface>Volume
Main Menu>Solution>MultiField Set Up>Interface>Volume

MGEN, *ITIME*, *INC*, *NODE1*, *NODE2*, *NINC*

Generates additional MDOF from a previously defined set.

SOLUTION: Master DOF

MP ME ST <> <> PR <> <> <> PP ED

ITIME, *INC*
Do this generation operation a total of *ITIME*s, incrementing all nodes in the set by *INC* each time after the first. *ITIME* must be > 1 for generation to occur. All previously defined master degree of freedom directions are included in the set. A component name may also be substituted for *ITIME*.

NODE1, *NODE2*, *NINC*
Generate master degrees of freedom from set beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and set is all selected nodes [**NSEL**]. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Generates additional master degrees of freedom from a previously defined set. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Master DOFs>User Selected>Copy
Main Menu>Solution>Master DOFs>User Selected>Copy

MITER, *NEL1*, *NEL2*, *RAD*, *NDIV*, *ESTRT*, *EINC*

Defines a mitered bend in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

NEL1, *NEL2*
Element numbers of the two intersecting straight pipes. Defaults to the last two straight pipe elements nearest the intersection of the last two runs.

RAD
Bend radius. If LR, use long radius standard (1.5 x OD) (default). If SR, use short radius standard (1.0 x OD).

NDIV
Number of divisions (elements) along bend (defaults to 2). A node is generated at the end of each division.

ESTRT
Number to be assigned to first element of bend (defaults to MAXEL + 1).

EINC
Element number increment (defaults to 1).

Notes

Defines a mitered bend of piecewise straight pipe elements (PIPE16) in place of the intersection of two previously defined straight pipe elements [**RUN**]. This command is similar to the **BEND** command except that straight pipe elements are used to form the bend instead of curved (elbow) elements.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Define Pipes>Miter

---

MLIST, *NODE1*, *NODE2*, *NINC*

Lists the MDOF of freedom.

SOLUTION: Master DOF

MP ME ST <> <> PR <> <> <> PP ED

NODE1, *NODE2*, *NINC*

List master degrees of freedom from *NODE1* to *NODE2* (defaults to*NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL (default), *NODE2* and *NINC* are ignored and masters for all selected nodes [**NSEL**] are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Notes

Lists the master degrees of freedom. Master degrees of freedom generated from the **TOTAL** command cannot be listed until after the first load step.

Menu Paths

Main Menu>Preprocessor>Loads>Master DOFs>User Selected>List All
Main Menu>Preprocessor>Loads>Master DOFs>User Selected>List Picked
Main Menu>Solution>Master DOFs>User Selected>List All
Main Menu>Solution>Master DOFs>User Selected>List Picked
Utility Menu>List>Other>Master DOF>At All Nodes
Utility Menu>List>Other>Master DOF>At Picked Nodes

MMF

Calculates the magnetomotive force along a path.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

Notes

MMF invokes an ANSYS macro which calculates the magnetomotive force (mmf) along a predefined path [**PATH**]. It is valid for both 2-D and 3-D magnetic field analyses. The calculated mmf value is stored in the parameter MMF.

A closed path [**PATH**], passing through the magnetic circuit for which mmf is to be calculated, must be defined before this command is issued. A counterclockwise ordering of points on the **PPATH** command will yield the correct sign on the mmf. The mmf is based on Ampere's Law. The macro makes use of calculated values of field intensity (H), and uses path operations for the calculations. All path items are cleared upon completion. The **MMF** macro sets the "ACCURATE" mapping method and "MAT" discontinuity option of the **PMAP** command.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>MMF

MODE, *MODE*, *ISYM*

Specifies the harmonic loading term for this load step.

SOLUTION: Load Step Options

MP ME ST <> <> <> <> <> <> PP ED

MODE

Number of harmonic waves around circumference for this harmonic loading term (defaults to 0).

ISYM

Symmetry condition for this harmonic loading term (not used when *MODE* = 0):

1

Symmetric (UX, UY, ROTZ, TEMP use cosine terms; UZ uses sine term) (default).

1

Antisymmetric (UX, UY, ROTZ, temp use sine terms; UZ uses cosine term).

Command Default

MODE = 0, *ISYM* = 1.

Notes

Used with axisymmetric elements having nonaxisymmetric loading capability (e.g., PLANE25, SHELL61, FLUID81, etc.). For analysis types **ANTYPE**,MODAL, HARMIC, TRANS, and SUBSTR, the term must be defined in the first load step and may not be changed in succeeding load steps.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>For Harmonic Ele
Main Menu>Solution>Load Step Opts>Other>For Harmonic Ele

MODIFY, *SET*, *LSTEP*, *ITER*, *CUMIT*, *TIME*, *Ktitle*

Changes the listed values of the data in a set.

AUX3: Results Files

MP ME ST <> <> <> <> <> <> PP ED

SET

Set of data in results file to be modified.

LSTEP

The new load step number.

ITER

The new load substep number.

CUMIT

The new cumulative iteration.

TIME
The new time/frequency value.

Ktitle
Indicates if the set title should be modified.

0
Keep the original title.

1
Change the title to the title specified with the most current **/TITLE** command.

Notes

Use this command to change the listed values in a data set in a results file. Using this command does not change any actual model data; it affects only the values listed in the results file.

For example, if you start with the following results file:

```
SET   TIME/FREQ    LOAD STEP   SUBSTEP  CUMULATIVE
 1     1.0000          1          1         1
 first load set

 2     2.0000           2          1         2
 second load set

 3     3.0000           3          1         3
 third load set

 4     4.0000           4          1         4
 fourth load set
```

and you then issue the following commands:

```
/title, modified title for set number 3
modify,3,5,2,5,4.5,1
```

The modified results file would look like this:

```
SET   TIME/FREQ    LOAD STEP   SUBSTEP  CUMULATIVE
 1     1.0000          1          1         1
 first load set

 2     2.0000           2          1         2
 second load set

 3     4.5000           5          2         5
 modified title for set number 3

 4     4.0000           4          1         4
 fourth load set
```

Menu Paths

Main Menu>Preprocessor>Loads>Other>For Harmonic Ele
Main Menu>Solution>Other>For Harmonic Ele

MODMSH, *Lab*

Controls the relationship of the solid model and the FE model.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

Lab

Relationship key:

STAT

Gives status of command (default). This applies only to the CHECK option (no status is provided for the DETACH option).

NOCHECK

Deactivates the checking of the solid model and the finite element model. Allows elements and nodes generated with the mesh commands to be modified directly (**EMODIF**, **NMODIF**, **EDELE**, **NDELE**, etc.). Also deactivates solid model hierarchical checking so that areas attached to volumes may be deleted etc.

Warning: Use of this command allows the solid model data base to be corrupted by subsequent operations.

CHECK

Reactivates future checking of the solid model.

DETACH

Releases all associativity between the current solid model and finite element model. ANSYS deletes any element attributes that were assigned to the affected solid model entities through default attributes (that is, through the **TYPE**, **REAL**, **MAT**, **SECNUM**, and **ESYS** command settings and a subsequent meshing operation). However, attributes that were assigned directly to the solid model entities (via the **KATT**, **LATT**, **AATT**, and **VATT** commands) are not deleted.

Caution: Once used it is not possible to select or define finite element model items in terms of the detached solid model or to clear the mesh.

Notes

Affects the relationship of the solid model (keypoints, lines, areas, volumes) and the finite element model (nodes, elements, and boundary conditions).

Menu Paths

Main Menu>Preprocessor>Checking Ctrls>Model Checking

MODOPT, *Method*, *NMODE*, *FREQB*, *FREQE*, *PRMODE*, *Nrmkey*

Specifies modal analysis options.

SOLUTION: Dynamic Options

MP ME ST <> VT PR <> EH <> PP ED

Method

Mode extraction method to be used for the modal analysis.

LANB
Block Lanczos (default)

SUBSP
Subspace iteration

REDUC
Householder (reduced)

UNSYM
Unsymmetric matrix (cannot be followed by a subsequent spectrum analysis).

DAMP
Damped system (cannot be followed by a subsequent spectrum analysis).

QRDAMP
Damped system using QR algorithm (cannot be followed by a subsequent spectrum analysis).

SX
Solve with the Variational Technology.

NMODE
Number of modes to extract. The value can depend on the value supplied for *Method*. Defaults to the number of master DOFs when *Method* = REDUC. For the other methods, *NMODE* has no default and must be specified. When *Method* = SUBSP, only half of the total number of DOFs can be extracted, *that is,* if the model contains 100 DOFs, only 50 modes are extracted even if *NMODE* is set to 100.

Recommendation: When *Method* = REDUC, *NMODE* should be less than half of the number of master DOFs. When *Method* = SUBSP, *NMODE* should be less than half of the total number of DOFs.

FREQB
Beginning, or lower end, of frequency range of interest.

For *Method* = LANB, SUBSP, UNSYM, DAMP, and QRDAMP, *FREQB* also represents the first shift point for the eigenvalue iterations. *FREQB* defaults to -1.0 if zero or blank for SUBSP, UNSYM, and DAMP. Eigenvalue extraction is most accurate near the shift point; multiple shift points are used in the LANB, SUBSP, and QRDAMP methods. For SUBSP method, and for LANB, UNSYM, DAMP, and QRDAMP methods with a positive *FREQB*, eigenvalues are output beginning at the shift point and increase in magnitude. For UNSYM and DAMP methods with a negative *FREQB*, eigenvalues are output beginning at zero magnitude and increase.

FREQE
Ending, or upper end, of frequency range of interest. *FREQE* defaults to 1e8 for *Method* = LANB, SUBSP, and QRDAMP. The default for the other methods is to calculate all modes, regardless of their maximum frequency.

PRMODE
Number of reduced modes to print for *Method* = REDUC.

Nrmkey
Mode shape normalization key:

OFF
Normalize the mode shapes to the mass matrix (default).

ON
Normalize the mode shapes to unity instead of to the mass matrix. If a subsequent spectrum or mode superposition analysis is planned, the mode shapes should be normalized to the mass matrix (that is, *Nrmkey* = OFF).

Notes

Specifies modal analysis (**ANTYPE**,MODAL) options. Additional options used only for subspace iteration eigenvalue extraction are specified by **SUBOPT**. Specifying the subspace option along with the PCG solver [**EQSLV**,PCG] is the same as choosing the Power Dynamics option on the GUI. If used in SOLUTION, this command is valid only within the first load step.

The Block Lanczos method, which is the default, is strongly recommended for high-frequency magnetic eigenvalue problems. The initial frequency guess is not critical. The ratio of FREQE to FREQB can be up to 1 x 10^6.

This command is also valid in PREP7.

Product Restrictions:

The damped, unsymmetric, and QR damped methods (DAMP, UNSYM, QRDAMP) are not available in the ANSYS Professional program. Also, for modal analyses using Variational Technology, *Method* = SX and *Nrmkey* = OFF are the only options.

Menu Paths

Main Menu>DesignXplorer VT>Solution>Solve
Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

MONITOR, *VAR*, *Node*, *Lab*

Controls contents of three variable fields in nonlinear solution monitor file.

SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> <> PP ED

VAR
One of three variable field numbers in the monitor file whose contents can be specified by the Lab field. Valid arguments are integers 1, 2, or 3. See Notes section for default values.

Node
The node number for which information is monitored in the specified *VAR* field. In the GUI, if *Node* = P, graphical picking is enabled. If blank, the monitor file lists the maximum value of the specified quantity (Lab field) for the entire structure.

Lab
The solution quantity to be monitored in the specified *VAR* field. Valid labels for solution quantities are UX, UY, and UZ (displacements); ROTX, ROTY, and ROTZ (rotations); and TEMP (temperature). Valid labels for reaction force are FX, FY, and FZ (structural force) and MX, MY, and MZ (structural moment). Valid label for heat flow rate is HEAT. For defaults see the Notes section.

Notes

This command is only active when **SOLCONTROL**,ON.

The monitor file is an ASCII file which is automatically created and saved when **SOLCONTROL** is active (ON). The monitor file always has an extension of **.mntr**, and takes its file name from the specified **Jobname**. If no **Jobname** is specified, the file name defaults to **file**.

You must issue this command once for each solution quantity you want to monitor at a specified node at each load step. You cannot monitor a reaction force during a linear analysis. The variable field contents can be redefined at each load step by reissuing the command. The monitored quantities are appended to the file for each load step.

The following example shows the format of a monitor file. Note that the file only records the solution substep history when a substep is convergent.

```
SOLUTION HISTORY INFORMATION FOR JOB: file.mntr

ANSYS Release 8.1                     10:12:43    05/15/2000

LOAD SUB- NO.   NO.  TOTL INCREMENT    TOTAL       VARIAB 1  VARIAB 2    VARIAB 3
STEP STEP ATTMP ITER ITER TIME/LFACT   TIME/LFACT  MONITOR   MONITOR     MONITOR
                                                      UZ        MZ         MxPl

 1    1    1     3     3   0.25000      0.25000    1.4145    0.19076E-06 0.78886E-30
 1    2    1     2     5   0.25000      0.50000    2.8283    0.92989E-06 0.78886E-30
 1    3    1     2     7   0.37500      0.87500    4.9467    0.33342E-05 0.78886E-30
 1    4    1     2     9   0.12500      1.0000     5.6519    0.16826E-05 0.78886E-30
 2    1    1     6    15   0.20000E-03  1.0002     4.2198  515.23        0.78886E-30
 2    2    2     6    26   0.10000E-03  1.0003     4.4849  593.03        0.78886E-30
 2    3    1     3    29   0.10000E-03  1.0004     4.7531  611.45        0.78886E-30
 2    4    1     3    32   0.15000E-03  1.0006     5.0696  621.83        0.78886E-30
 2    5    1     4    36   0.22500E-03  1.0008     5.4428  628.42        0.78886E-30
 2    6    1     4    40   0.33750E-03  1.0011     5.8928  632.78        0.78886E-30
 2    7    1     5    45   0.50625E-03  1.0016     6.4454  635.62        0.78886E-30
 2    8    1     7    52   0.75938E-03  1.0024     7.1375  637.22        0.78886E-30
 2    9    1     5    57   0.75938E-03  1.0031     7.7422  637.66        0.78886E-30
 2   10    1     6    63   0.11391E-02  1.0043     8.5588  637.42        0.78886E-30
 2   11    2     3    72   0.76887E-03  1.0050     9.0721  636.96        0.78886E-30
 2   12    1     3    75   0.76887E-03  1.0058     9.5648  636.35        0.78886E-30
 2   13    1     3    78   0.11533E-02  1.0070    10.277   635.25        0.78886E-30
 2   14    1     4    82   0.17300E-02  1.0087    11.306   633.37        0.78886E-30
 2   15    1     6    88   0.25949E-02  1.0113    12.802   630.21        0.78886E-30
 2   16    1     5    93   0.25949E-02  1.0139    14.273   626.81        0.78886E-30
 2   17    1     7   100   0.38924E-02  1.0178    16.477   621.42        0.78886E-30
 2   18    1     6   106   0.38924E-02  1.0217    18.704   615.77        0.78886E-30
 2   19    2     4   116   0.26274E-02  1.0243    20.229   611.83        0.78886E-30
 2   20    1     4   120   0.26274E-02  1.0269    21.777   607.80        0.78886E-30
```

The following details the contents of the various fields in the monitor file:

LOAD STEP
: The current load step number.

SUBSTEP
: The current substep (time step) number.

NO. ATTEMPT
: The number of attempts made in solving the current substep. This number is equal to the number of failed attempts (bisections) plus one (the successful attempt).

NO. ITER
: The number of iterations used by the last successful attempt.

TOTL. ITER
: Total cumulative number of iterations (including each iteration used by a bisection).

INCREMENT

TIME/LFACT
Time or load factor increments for the current substep.

TOTAL TIME/LFACT
Total time (or load factor) for the last successful attempt in the current substep.

VARIAB 1
Variable field 1. In this example, the field is reporting the UZ value. By default, this field lists the CPU time used up to (but not including) the current substep.

VARIAB 2
Variable field 2. In this example, the field is reporting the MZ value. By default, this field lists the maximum displacement in the entire structure.

VARIAB 3
Variable field 3. By default (and in the example), this field reports the maximum equivalent plastic strain in the entire structure.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Monitor
Main Menu>Solution>Load Step Opts>Nonlinear>Monitor

---

*MOONEY, *STRAIN*, *STRESS*, --, *CONST*, *CALC*, *SORTSN*, *SORTSS*, *Fname*, *Ext*

Calculates Mooney-Rivlin hyperelastic constants from test data.

PREP7: Materials

MP ME ST <> <> <> <> <> <> PP ED

STRAIN
Name of the array parameter containing the laboratory strain data. You must have previously filled this array with *engineering* strain values (no other measure of strain is valid) before you execute ***MOONEY**. If Uniaxial data (tension or compression) are to be used, they must be placed in the first column of this array. If Equibiaxial data (tension or compression) are to be used, they must be placed in the second column of this array. If Shear data are to be used, they must be placed in the third column of this array. If any test type is not used, the corresponding column of the *STRAIN* array should be left blank.

STRESS
Name of the array parameter containing the laboratory stress data. You must have previously filled the *STRESS* array with *engineering* stress values (no other measure of stress is valid) before you execute ***MOONEY**. The stress values must be placed in the *STRESS* array in locations corresponding to the locations of the companion strain values in the *STRAIN* array.

--
Unused field

CONST
Name of the array parameter vector to which the hyperelastic material constants will be written. The *CONST* array must have been previously defined [***DIM**] to have a dimension of either 2, 5, or 9 (corresponding to a two-term, five-term, or nine-term Mooney-Rivlin material model). Using any dimension other than 2, 5, or 9 for the *CONST* vector array will cause an error message to be generated.

CALC
Name of the array parameter vector in which calculated engineering stress values determined from the Mooney-Rivlin constants will be placed. For this and the following two parameters (SORTSN, SORTSS), column 1 contains uniaxial data, column 2 contains equibiaxial data, and column 3 contains shear data.

SORTSN
Name of the array parameter vector in which the sorted experimental strain data will be placed.

SORTSS
Name of the array parameter vector in which the sorted laboratory test data will be placed.

Fname
Text file name and directory path (248 characters maximum, including directory) to which the determined constants will be written (in the form of **TBDATA** commands). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to TB if *Fname* is blank.

Notes

Note that the use of the ***MOONEY** command for fitting data to the Mooney-Rivlin hyperelastic model is discouraged in favor of the general curve fitting method provided by the **TBFT** command. SeeChapter 9, "Material Curve Fitting" in the *ANSYS Structural Analysis Guide* for details.

For ***MOONEY**, the *array parameters* *STRAIN*, *STRESS*, *CALC*, *SORTSN*, and *SORTSS* must have been previously defined [***DIM**] to have dimensions *N*x3, where*N* is the maximum number of data points in any one of the three basic test types (uniaxial, equibiaxial, and planar or pure shear).

MOONEY** calculates the Mooney-Rivlin hyperelastic material constants from laboratory stress-strain test data. Once the program determines these constants, it writes them to three places: to the database (in memory), to the array parameter *CONST*, and to a text file (in the form of **TBDATA** commands). You can use the ***EVAL** command to check the quality of the resulting material properties. You must have previously dimensioned [DIM**] all array parameters used by ***MOONEY**, and you must also have set *LAB* = MOONEY and *TBOPT* = 1 on the **TB** command, before you execute ***MOONEY**.

Up to three different types of laboratory stress-strain tests can be used (in any combination):

- Uniaxial (tension or compression)
- Equibiaxial (tension or compression)
- Shear (Planar Tension or Planar Compression)

The ***MOONEY** command cannot be used with the Mooney-Rivlin material model for explicit dynamic analysis in ANSYS LS-DYNA. To use experimental Mooney-Rivlin data for an explicit dynamic analysis, use *TBOPT* = 2 on the **TB** command (see Nonlinear Elastic Models in the *ANSYS/LS-DYNA User's Guide* for more details).

Menu Paths

This command cannot be accessed from a menu.

MOPT, *Lab*, *Value*

Specifies meshing options.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

Lab

Meshing option to be specified (dictates the meaning of *Value*):

AORDER

Mesh by ascending area size order. Set *Value* to ON to mesh smaller areas first. Using this results in finer meshes in critical areas for volume meshes; this can be used for cases where **SMRTSIZE** does not mesh as needed. Default is OFF.

EXPND

Area mesh expansion (or contraction) option. (This option is the same as **SMRTSIZE**,,,EXPND.) This option is used to size internal elements in an area based on the size of the elements on the area's boundaries. *Value* is the expansion (or contraction) factor. For example, issuing **MOPT**,EXPND,2 before meshing an area will allow a mesh with elements that are approximately twice as large in the interior of an area as they are on the boundary. If *Value* is less than 1, a mesh with smaller elements on the interior of the area will be allowed. *Value* for this option should be greater than 0.5 but less than 4. *Value* defaults to 1, which does not allow expansion or contraction of internal element sizes (except when using AESIZE sizing). If *Value* = 0, the default value of 1 will be used. The actual size of the internal elements will also depend on the TRANS option, or upon AESIZE or ESIZE sizing, if used.

TETEXPND

Tet-mesh expansion (or contraction) option. This option is used to size internal elements in a volume based on the size of the elements on the volume's boundaries. *Value* is the expansion (or contraction) factor. For example, issuing **MOPT**,TETEXPND,2 before meshing a volume will allow a mesh with elements that are approximately twice as large in the interior of the volume as they are on the boundary. If *Value* is less than 1, a mesh with smaller elements on the interior of the volume will be allowed. *Value* for this option should be greater than 0.1 but less than 3. *Value* defaults to 1, which does not allow expansion or contraction of internal element sizes. If *Value* = 0, the default value of 1 will be used. If *Value* is greater than 2, mesher robustness may be affected. The TETEXPND option is supported for both the **VMESH** and **FVMESH** commands. Tet-mesh expansion is the only mesh control supported by **FVMESH**.

TRANS

Mesh transition option. (This option is the same as **SMRTSIZE**,,,,TRANS.) This option is used to control how rapidly elements are permitted to change in size from the boundary to the interior of an area. *Value* is the transitioning factor. *Value* defaults to 2.0, which permits elements to approximately double in size as they approach the interior of the area. (If *Value* = 0, the default value of 2 will be used.) *Value* must be greater than 1 and, for best results, should be less than 4. The actual size of the internal elements will also depend on the EXPND option, or upon AESIZE or ESIZE sizing, if used.

AMESH

Triangle surface meshing option. Valid inputs for *Value* are:

DEFAULT

Let ANSYS choose which triangle mesher to use. In most cases, ANSYS will choose the main triangle mesher, which is the Riemann space mesher. If the chosen mesher fails for any reason, ANSYS invokes the alternate mesher and retries the meshing operation.

MAIN

ANSYS uses the main triangle mesher (Riemann space mesher), and it does not invoke an alternate mesher if the main mesher fails. The Riemann space mesher is well suited for most surfaces.

ALTERNATE
ANSYS uses the first alternate triangle mesher (3-D tri-mesher), and it does not invoke another mesher if this mesher fails. This option is not recommended due to speed considerations. However, for surfaces with degeneracies in parametric space, this mesher often provides the best results.

ALT2
ANSYS uses the second alternate triangle mesher (2-D parametric space mesher), and it does not invoke another mesher if this mesher fails. This option is not recommended for use on surfaces with degeneracies (spheres, cones, etc.) or poorly parameterized surfaces because poor meshes may result.

QMESH
Quadrilateral surface meshing option. (Quadrilateral surface meshes will differ based on which triangle surface mesher is selected. This is true because all free quadrilateral meshing algorithms use a triangle mesh as a starting point.) Valid inputs for *Value* are:

DEFAULT
Let ANSYS choose which quadrilateral mesher to use. In most cases, ANSYS will choose the main quadrilateral mesher, which is the Q-Morph (quad-morphing) mesher. For very coarse meshes, ANSYS may choose the alternate quadrilateral mesher instead. In most cases, the Q-Morph mesher results in higher quality elements. If either mesher fails for any reason, ANSYS invokes the other mesher and retries the meshing operation. (Default.)

MAIN
ANSYS uses the main quadrilateral mesher (Q-Morph mesher), and it does not invoke the alternate mesher if the main mesher fails.

ALTERNATE
ANSYS uses the alternate quadrilateral mesher, and it does not invoke the Q-Morph mesher if the alternate mesher fails. To use the alternate quadrilateral mesher, you must also select **MOPT**,AMESH,ALTERNATE or **MOPT**,AMESH,ALT2.

VMESH
Tetrahedral element meshing option. Valid inputs for *Value* are:

DEFAULT
Let ANSYS choose which tetrahedra mesher to use. ANSYS always uses the alternate tetrahedra mesher when meshing with p-elements. Otherwise, it usually uses the main tetrahedra mesher.

MAIN
Use the main tetrahedra mesher (Delaunay technique mesher). (GHS3D meshing technology by P. L. George, INRIA, France.) For most models, this mesher is significantly faster than the alternate mesher.

ALTERNATE
Use the alternate tetrahedra mesher (advancing front mesher). This mesher is the ANSYS Revision 5.2 tetrahedra mesher. It does not support the generation of a tetrahedral volume mesh from facets [**FVMESH**]. If this mesher is selected and you issue the **FVMESH** command, ANSYS uses the main tetrahedra mesher to create the mesh from facets and issues a warning message to notify you.

SPLIT
Quad splitting option for non-mapped meshing. If *Value* = 1, ON, or ERR, quadrilateral elements in violation of shape error limits are split into triangles (default). If *Value* = 2 or WARN, quadrilateral elements in violation of either shape error *or* warning limits are split into triangles. If *Value* = OFF, splitting does not occur, regardless of element quality.

LSMO

Line smoothing option. *Value* can be ON or OFF. If *Value* = ON, smoothing of nodes on area boundaries is performed during smoothing step of meshing. During smoothing, node locations are adjusted to achieve a better mesh. If *Value* = OFF (default), no smoothing takes place at area boundaries.

CLEAR

This option affects the element and node numbering after clearing a mesh. If *Value* = ON (default), the starting node and element numbers will be the lowest available number after the nodes and elements are cleared. If *Value* = OFF, the starting node and element numbers are not reset after the clear operation (which was the default behavior for ANSYS versions prior to Release 5.3).

PYRA

Transitional pyramid elements option. *Value* can be ON or OFF. If *Value* = ON (default), ANSYS automatically creates transitional pyramid elements, when possible. Pyramids may be created at the interface of tetrahedral and hexahedral elements, or directly from quadrilateral elements. For pyramids to be created, you must also issue the command **MSHAPE**,1,3D (degenerate 3-D elements). If *Value* = OFF, ANSYS does not create transitional pyramid elements.

TIMP

Identifies the level of tetrahedra improvement to be performed when the next free volume meshing operation is initiated [**VMESH**, **FVMESH**]. (For levels 2-5, improvement occurs primarily through the use of face swapping and node smoothing techniques.) Valid inputs for *Value* are:

0

Turn off tetrahedra improvement. Although this value can lead to faster tetrahedral mesh creation, it is *not recommended* because it often leads to poorly shaped elements and mesh failures.

1

Do the minimal amount of tetrahedra improvement. (Default.) This option is supported by the main tetrahedra mesher only [**MOPT**,VMESH,MAIN]. If the alternate tetrahedra mesher [**MOPT**,VMESH,ALTERNATE] is invoked with this setting, ANSYS automatically performs tetrahedra improvement at level 3 instead [**MOPT**,TIMP,3].

2

Perform the least amount of swapping/smoothing. No improvement occurs if all tetrahedral elements are within acceptable limits.

3

Perform an intermediate amount of swapping/smoothing. Some improvement is always done.

4

Perform the greatest amount of swapping/smoothing. Meshing takes longer with this level of improvement, but usually results in a better mesh.

5

Perform the greatest amount of swapping/smoothing, plus additional improvement techniques. This level of improvement usually produces results that are similar to those at level 4, except for very poor meshes.

6

For linear tetrahedral meshes, this value provides the same level of improvement as **MOPT**,TIMP,5. For quadratic tetrahedral meshes, this value provides an additional pass of cleanup. This value is supported for both the main [**MOPT**,VMESH,MAIN] and alternate [**MOPT**,VMESH,ALTERNATE] tetrahedra meshers.

STAT

Display status of **MOPT** settings. *Value* is ignored.

DEFA
Set all **MOPT** options to default values. *Value* is ignored.

Value
Value, as described for each different *Lab* above.

Notes

See the *ANSYS Modeling and Meshing Guide* for more information on the **MOPT** command and its options.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesher Opts
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Area Cntrls
Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Volu Cntrls
Utility Menu>List>Status>Preprocessor>Solid Model

MORPH, *Option*

Turns morphing on or off.

PREP7: Morphing

MP <> <> <> <> <> <> <> <> PP ED

Option

ON
Turns on morphing for field elements.

OFF
Turns off morphing for field elements.

Notes

MORPH is applicable to any non-structural field analysis. It activates displacement degrees of freedom for non-structural elements so that boundary conditions may be placed on the field mesh to constrain the movement of the non-structural mesh during morphing. It morphs the non-structural mesh using displacements transferred at the surface interface between the structural field and the non-structural field.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Element Morphing
Main Menu>Solution>Load Step Opts>Other>Element Morphing

MOVE, *NODE*, *KC1*, *X1*, *Y1*, *Z1*, *KC2*, *X2*, *Y2*, *Z2*

Calculates and moves a node to an intersection.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

NODE
Move this node. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

KC1
First coordinate system number. Defaults to 0 (global Cartesian).

X1, *Y1*, *Z1*
Input one or two values defining the location of the node in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Fields are R1, θ1, Z1 for cylindrical, or R1, θ1, Φ1 for spherical or toroidal.

KC2
Second coordinate system number.

X2, *Y2*, *Z2*
Input two or one value(s) defining the location of the node in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Fields are R2, θ2, Z2 for cylindrical, or R2, θ2, Φ2 for spherical or toroidal.

Notes

Calculates and moves a node to an intersection location. The node may have been previously defined (at an approximate location) or left undefined (in which case it is internally defined at the **SOURCE** location). The actual location is calculated from the intersection of three surfaces (implied from three coordinate constants in two different coordinate systems). The three (of six) constants easiest to define should be used. The program will calculate the remaining three coordinate constants. All arguments, except *KC1*, must be input. Use the repeat command [***REPEAT**] after the **MOVE** command to define a line of intersection by repeating the move operation on all nodes of the line.

Surfaces of constant value are implied by some commands by specifying a single coordinate value. Implied surfaces are used with various commands [**MOVE**, **KMOVE**, **NSEL**, etc.]. Three surfaces are available with each of the four coordinate system types. Values or X, Y, or Z may be constant for the Cartesian coordinate system; values of R, θ, or Z for the cylindrical system; and values of R, θ, Φ for the spherical and toroidal systems. For example, an X value of 3 represents the Y-Z plane (or surface) at X=3. In addition, the parameters for the cylindrical and spherical coordinate systems may be adjusted [**CS**, **LOCAL**] to form elliptical surfaces. For surfaces in elliptical coordinate systems, a surface of "constant" radius is defined by the radius value at the X-axis. Surfaces of constant value may be located in local coordinate systems [**LOCAL**, **CLOCAL**, **CS**, or **CSKP**] to allow for any orientation.

The intersection calculation is based on an iterative procedure (250 iterations maximum) and a tolerance of 1.0E-4. The approximate location of a node should be sufficient to determine a unique intersection if more than one intersection point is possible. Tangent "intersections" should be avoided. If an intersection is not found, the node is placed at the last iteration location.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>To Intersect

MP, *Lab*, *MAT*, *C0*, *C1*, *C2*, *C3*, *C4*

Defines a linear material property as a constant or a function of temperature.

PREP7: Materials

MP ME ST DY <> PR EM EH FL PP ED

Lab

Valid material property label. Applicable labels are listed under "Material Properties" in the input table for each element type in the *ANSYS Elements Reference*. See Section 2.4: Linear Material Properties of the *ANSYS Elements Reference* for more complete property label definitions:

EX
Elastic moduli (also EY, EZ).

ALPX
Secant coefficients of thermal expansion (also ALPY, ALPZ).

CTEX
Instantaneous coefficients of thermal expansion (also CTEY, CTEZ).

THSX
Thermal strain (also THSY, THSZ).

REFT
Reference temperature. Must be defined as a constant; C1 through C4 are ignored.

PRXY
Major Poisson's ratios (also PRYZ, PRXZ).

NUXY
Minor Poisson's ratios (also NUYZ, NUXZ).

GXY
Shear moduli (also GYZ, GXZ).

DAMP
K matrix multiplier for damping.

Note: If used in an explicit dynamic analysis, the value corresponds to the percentage of damping in the high frequency domain. For example, 0.1 roughly corresponds to 10% damping in the high frequency domain.

DMPR
Constant material damping coefficient.

MU
Coefficient of friction.

DENS
Mass density.

C
Specific heat.

ENTH
Enthalpy.

KXX
Thermal conductivities (also KYY, KZZ).

HF
Convection or film coefficient.

EMIS
Emissivity.

QRATE
Heat generation rate.

VISC
Viscosity.

SONC
Sonic velocity.

RSVX
Electrical resistivities (also RSVY, RSVZ).

PERX
Electric relative permittivities (also PERY, PERZ).

Note — If you enter permittivity values less than 1 for SOLID5, PLANE13, or SOLID98, the program interprets the values as absolute permittivity. Values input for PLANE223, SOLID226, or SOLID227 are always interpreted as relative permittivity.

MURX
Magnetic relative permeabilities (also MURY, MURZ).

MGXX
Magnetic coercive forces (also MGYY, MGZZ).

LSST
Dielectric loss tangent.

MAT
Material reference number to be associated with the elements (defaults to the current MAT setting [**MAT**]).

C0
Material property value, or if a property-versus-temperature polynomial is being defined, the constant term in the polynomial.

C1, *C2*, *C3*, *C4*
Coefficients of the linear, quadratic, cubic, and quartic terms, respectively, in the property-versus-temperature polynomial. Leave blank (or set to zero) for a constant material property.

Notes

MP defines a linear material property as a constant or in terms of a fourth order polynomial as a function of temperature. See the **TB** command for nonlinear material property input. Linear material properties typically require a single substep for solution, whereas nonlinear material properties require multiple substeps; see Section 2.4: Linear Material Properties of the *ANSYS Elements Reference* for details.

If the constants C1 - C4 are input, the polynomial

$$\text{Property} = C0 + C1(\text{T}) + C2(\text{T})^2 + C3(\text{T})^3 + C4(\text{T})^4$$

is evaluated at discrete temperature points with linear interpolation between points (i.e., piecewise linear representation) and a constant-valued extrapolation beyond the extreme points. The **MPTEMP** or **MPTGEN** commands

must be used for second and higher order properties to define appropriate temperature steps. First-order properties use two discrete points (±9999°).

A polynomial input is not valid in an explicit dynamic analysis or for *Lab* = DAMP. C1, C2, C3, and C4 are ignored.

This command is also valid in SOLUTION.

Product Restrictions:

In ANSYS without Emag enabled, the MUR_ and MG__ properties are not allowed. In ANSYS Professional, all structural and thermal properties are allowed except DAMP and MU. In ANSYS Emag, only the RSV_, PER_, MUR_, and MG__ properties are allowed. The LSST property is available only for products that include ANSYS Emag, and can be used only in high-frequency analyses.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

MPAMOD, *MAT*, *DEFTEMP*

Modifies temperature-dependent secant coefficients of thermal expansion.

PREP7: Materials

MP ME ST <> <> PR EM <> FL PP ED

MAT
Material number for which the secant coefficients of thermal expansion (SCTE's) are to be modified. Defaults to 1.

DEFTEMP
Definition temperature at which the existing SCTE-versus-temperature tables were defined. Defaults to zero.

Notes

This command converts temperature-dependent SCTE data (properties ALPX, ALPY, ALPZ) from the definition temperature (*DEFTEMP*) to the reference temperature defined by **MP**,REFT or **TREF**. If both the **MP**,REFT and **TREF** commands have been issued, the reference temperature defined by the **MP**,REFT command will be used.

This command does not apply to the instantaneous coefficients of thermal expansion (properties CTEX, CTEY, CTEZ) or to the thermal strains (properties THSX, THSY, THSZ).

See Section 2.4: Linear Material Properties of the *ANSYS Elements Reference* and the *ANSYS, Inc. Theory Reference* for more details.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Convert ALPx
Main Menu>Preprocessor>Material Props>Convert ALPx

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Convert ALPx

---

MPCHG, *MAT*, *ELEM*

Changes the material number attribute of an element.

PREP7: Materials
SOLUTION: Misc Loads

MP ME ST DY <> PR EM <> FL PP ED

MAT
Assign this material number to the element. Material numbers are defined with the material property commands [**MP**].

ELEM
Element for material change. If ALL, change materials for all selected elements [**ESEL**].

Notes

Changes the material number of the specified element. Between load steps in SOLUTION, material properties cannot be changed from linear to nonlinear, or from one nonlinear option to another.

If you change from one MKIN model to another MKIN model, the different MKIN models need to have the same number of data points. This requirement also applies if you change from one KINH model to another KINH model, or from one CHABOCHE model to another CHABOCHE model.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Change Mat Num
Main Menu>Preprocessor>Material Props>Change Mat Num
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Change Mat Num

---

MPCOPY, --, *MATF*, *MATT*

Copies linear material model data from one material reference number to another.

PREP7: Materials

MP ME ST DY <> PR EM <> <> PP ED

--
Unused field

MATF
Material reference number from where material property data will be copied.

MATT
Material reference number to where material property data will be copied.

Notes

The **MPCOPY** command copies linear material properties only, which are all properties defined through the **MP** command. If you copy a model that includes both linear and yield behavior constants (for example, a BKIN

model), the **MPCOPY** and **TBCOPY**, ALL commands are used together to copy the entire model. All input data associated with the model is copied, that is, all data defined through the **MP** and **TB** commands.

Also, if you copy a material model using the Material Model Interface (**Edit> Copy**), both the commands **MPCOPY** and **TBCOPY**, ALL are issued, regardless of whether the model includes linear constants only, or if it includes a combination of linear and yield behavior constants.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

MPDATA, *Lab*, *MAT*, *STLOC*, *C1*, *C2*, *C3*, *C4*, *C5*, *C6*

Defines property data to be associated with the temperature table.

PREP7: Materials

MP ME ST DY <> PR EM EH FL PP ED

Lab

Valid property label. Applicable labels are listed under "Material Properties" in the input table for each element type in the *ANSYS Elements Reference*. See Section 2.4: Linear Material Properties of the *ANSYS Elements Reference* for more complete property label definitions:

EX
Elastic moduli (also EY, EZ).

ALPX
Secant coefficients of thermal expansion (also ALPY, ALPZ). (See also **MPAMOD** command for adjustment to reference temperature).

CTEX
Instantaneous coefficients of thermal expansion (also CTEY, CTEZ).

THSX
Thermal strain (also THSY, THSZ).

REFT
Reference temperature (may not be temperature dependent).

PRXY
Major Poisson's ratios (also PRYZ, PRXZ).

NUXY
Minor Poisson's ratios (also NUYZ, NUXZ).

GXY
Shear moduli (also GYZ, GXZ).

DAMP
K matrix multiplier for damping.

DMPR
Constant material damping coefficient.

MU
Coefficient of friction.

DENS
Mass density.

C
Specific heat.

ENTH
Enthalpy.

KXX
Thermal conductivities (also KYY, KZZ).

HF
Convection or film coefficient.

EMIS
Emissivity.

QRATE
Heat generation rate.

VISC
Viscosity.

SONC
Sonic velocity.

RSVX
Electrical resistivities (also RSVY, RSVZ).

PERX
Electric relative permittivities (also PERY, PERZ).

MURX
Magnetic relative permeabilities (also MURY, MURZ).

MGXX
Magnetic coercive forces (also MGYY, MGZZ).

LSST
Dielectric loss tangent.

The **MPDATA** command may also be used to enter temperature dependent properties for fluids in a CFD analysis with FLOTRAN via FLUID141 and FLUID142. Valid **MPDATA** labels for a CFD analysis in a non-solid region are:

DENS
Density of fluid. This is the same as the label used to specify mass density with the **FLDATA** command.

C
Specific heat of fluid. This is equivalent to the SPHT label used to specify conductivity with the **FLDATA** command.

KXXX
Thermal conductivity of fluid.

VISC
Viscosity of fluid. This is the same as the label used to specify kinematic velocity with the **FLDATA** command.

MAT
Material reference number to be associated with the elements (defaults to 1 if you specify zero or no material number).

STLOC
Starting location in table for generating data. For example, if *STLOC* = 1, data input in the *C1* field is the first constant in the table. If *STLOC* = 7, data input in the *C1* field is the seventh constant in the table, etc. Defaults to the last location filled + 1.

C1, *C2*, *C3*, *C4*, *C5*, *C6*
Property data values assigned to six locations starting with *STLOC*. If a value is already in this location, it is redefined. A blank (or zero) value for *C1* resets the previous value in *STLOC* to zero. A value of zero can only be assigned by *C1*. Blank (or zero) values for *C2* to *C6* leave the corresponding previous values unchanged.

Notes

Defines a table of property data to be associated with the temperature table. Repeat **MPDATA** command for additional values (100 maximum). Temperatures must be defined first [**MPTEMP**]. Also stores assembled property function table (temperature and data) in virtual space.

This command is also valid in SOLUTION.

Product Restrictions

In ANSYS without Emag enabled, the MUR_ and MG__ properties are not allowed. In ANSYS Professional, all structural and thermal properties are allowed except DAMP and MU. In ANSYS Emag, only the RSV_, PER_, MUR_, and MG__ properties are allowed. Only products that include ANSYS Emag can use the LSST property.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

MPDELE, *Lab*, *MAT1*, *MAT2*, *INC*

Deletes linear material properties.

PREP7: Materials

MP ME ST DY <> PR EM <> FL PP ED

Lab
Material property label (see **MP** command for valid labels). If ALL, delete properties for all applicable labels.

MAT1, *MAT2*, *INC*
Delete materials from *MAT1* to *MAT2* (defaults to *MAT1*) in steps of *INC* (defaults to 1). If *MAT1* = ALL, *MAT2* and *INC* are ignored and the properties for all materials are deleted.

Notes

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

MPDRES, *LabF*, *MATF*, *LabT*, *MATT*

Reassembles existing material data with the temperature table.

PREP7: Materials

MP ME ST <> <> PR EM <> FL PP ED

LabF
Material property label associated with *MATF*.

MATF
Material reference number of property to restore from virtual space.

LabT
Material property label associated with *MATT* (defaults to label associated with *MATF*).

MATT
Material reference number assigned to generated property (defaults to *MATF*).

Notes

Restores into the database (from virtual space) a data table previously defined [**MP**] for a particular property, assembles data with *current* database temperature table, and stores back in virtual space as a new property.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Other>Change Mat Props>Modify Temps
Main Menu>Preprocessor>Material Props>Modify Temps
Main Menu>Solution>Other>Change Mat Props>Modify Temps

/MPLIB, *R-W_opt*, *PATH*

Sets the default material library read and write paths.

PREP7: Materials

MP ME ST DY <> PR EM <> FL PP ED

R-W_opt
Determines what path is being set. Possible values are:

READ
Set the read path.

WRITE
Set the write path.

STAT
Report what read and write paths are currently in use.

PATH
The directory path to be used for material library files.

Notes

The **/MPLIB** command sets two path strings used in conjunction with the material library feature and the **MPREAD** and **MPWRITE** commands.

For **MPREAD**, when you use the *LIB* option and the directory portion of the specification for the material library file is blank, the command searches for the file in these locations: the current working directory, the user's home directory, the user-specified material library directory (as defined by the **/MPLIB**,READ,*PATH* command), and */ansys_dir*/**matlib**.

For **MPWRITE**, when you use the *LIB* option and the directory portion of the specification for the material library file is blank, the command writes the material library file to the directory specified by the **/MPLIB**,WRITE,*PATH* command (if that path has been set). If the path has not been set, the default is to write the file to the current working directory.

The Material Library files supplied with the distribution disks are meant for demonstration purposes only. These files are not intended for use in customer applications.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Lib Path Status
Main Menu>Preprocessor>Material Props>Material Library>Lib Path Status
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Lib Path Status

---

MPLIST, *MAT1*, *MAT2*, *INC*, *Lab*, *TEVL*

Lists linear material properties.

PREP7: Materials

MP ME ST DY <> PR EM <> FL PP ED

MAT1, *MAT2*, *INC*
List materials from *MAT1* to *MAT2* (defaults to *MAT1*) in steps of *INC* (defaults to 1). If *MAT1*= ALL (default), *MAT2* and *INC* are ignored and properties for all material numbers are listed.

Lab
Material property label (see the **MP** command for labels). If ALL (or blank), list properties for all labels. If EVLT, list properties for all labels evaluated at TEVL.

TEVL
Evaluation temperature for *Lab* = EVLT listing (defaults to **BFUNIF**).

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Properties>All Materials
Utility Menu>List>Properties>All Matls, All Temps
Utility Menu>List>Properties>All Matls, Specified Temp
Utility Menu>List>Properties>Specified Matl, All Temps

MPPLOT, *Lab*, *MAT*, *TMIN*, *TMAX*, *PMIN*, *PMAX*

Plots linear material properties as a function of temperature.

PREP7: Materials

MP ME ST <> <> PR EM <> FL PP ED

Lab
Linear material property label (EX, EY, etc.) [**MP**].

MAT
Material reference number. Defaults to 1.

TMIN
Minimum abscissa value to be displayed.

TMAX
Maximum abscissa value.

PMIN
Minimum property (ordinate) value to be displayed.

PMAX
Maximum property value.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Materials

MPREAD, *Fname*, *Ext*, --, LIB

Reads a file containing material properties.

PREP7: Materials

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify the LIB option, the default directory is the current working directory. If you specify the LIB option, the default is the

following search path: the current working directory, the user's home directory, MPLIB_DIR (as specified by the **/MPLIB**,READ,PATH command) and ***/ansys_dir*/matlib** (as defined by installation). If you use the default for your directory, you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

If you omit the default extension is MP. extension is units_MPL, where units is the system of units currently in use. (See the description of the **/UNITS** command.) For example, if **/UNITS** is set to SI, the extension defaults to SI_MPL.

--
Unused field

LIB
Reads material library files previously written with the **MPWRITE** command. (See the description of the LIB option for the **MPWRITE** command.) The only allowed value for LIB is LIB.

The LIB field indicates that the specified file was written by **MPWRITE** using the LIB option, and that the file is consistent with the material library file format. When the **MPREAD** command executes, the ANSYS program reads material properties defined in the specified file into the current ANSYS working database. The currently selected material, as defined by the **MAT** command (**MAT**,MAT), determines the material number used when reading the material properties. The LIB option for **MPREAD** and **MPWRITE** supports storing and retrieving both linear and nonlinear properties.

Notes

Material properties written to a file without the LIB option do not support nonlinear properties. Also, properties written to a file without the LIB option are restored in the same material number as originally defined. To avoid errors, use **MPREAD** with the LIB option only when reading files written using **MPWRITE** with the LIB option.

If you omit the LIB option for **MPREAD**, this command supports only linear properties.

Material numbers are hardcoded. If you write a material file without specifying the LIB option, then read that file in using the **MPREAD** command with the LIB option, the ANSYS program will not write the file to a new material number. Instead, it will write the file to the "old" material number (the number specified on the **MPWRITE** command that created the file.)

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Export Library
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Import Library
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Select Units
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Read from File
Main Menu>Preprocessor>Material Props>Material Library>Export Library
Main Menu>Preprocessor>Material Props>Material Library>Import Library

Main Menu>Preprocessor>Material Props>Material Library>Select Units
Main Menu>Preprocessor>Material Props>Read from File
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Export Library
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Import Library
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Select Units
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Read from File

---

MPRINT, *KEY*

Specifies that radiation matrices are to be printed.

AUX12: Radiation Substructures

MP ME ST <> <> PR <> <> <> PP ED

KEY
Print key:

0
Do not print matrices.

1
Print matrices.

Command Default

Matrices are not printed.

Notes

Specifies that the element and node radiation matrices are to be printed when the **WRITE** command is issued. If *KEY* = 1, form factor information for each element will also be printed.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Write Matrix

MPTEMP, *STLOC*, *T1*, *T2*, *T3*, *T4*, *T5*, *T6*

Defines a temperature table for material properties.

PREP7: Materials

MP ME ST DY <> PR EM <> FL PP ED

STLOC
Starting location in table for entering temperatures. For example, if *STLOC* = 1, data input in the *T1* field applies to the first constant in the table. If *STLOC* = 7, data input in the *T1* field applies to the seventh constant in the table, etc. Defaults to the last location filled + 1.

T1, *T2*, *T3*, *T4*, *T5*, *T6*
Temperatures assigned to six locations starting with *STLOC*. If a value is already in this location, it will be re-defined. A blank (or zero) value for *T1* resets the previous value in *STLOC* to zero. A value of zero can only be assigned by *T1*. Blank (or zero) values for *T2* to *T6* leave the corresponding previous values unchanged.

Command Default

No temperature table defined (i.e., properties must be defined as a constant or linear function of temperature with the **MP** command).

Notes

Defines a temperature table to be associated with the property data table [**MPDATA**]. These temperatures are also used for polynomial property evaluation, if defined [**MP**]. Temperatures must be defined in non-descending order. Issue **MATER** $ **STAT** to list the current temperature table. Repeat **MPTEMP** command for additional temperatures (100 maximum). If all arguments are blank, the temperature table is erased.

For clear definition, the temperature range you define with the **MPTEMP** command should include the entire range you'll use in subsequently defined materials. To assist the user in this, the first (and only the first) excursion out of the temperature range defined by the **MPTEMP** commands is flagged with a warning message. Similarly, the reference temperature (**TREF** or **MP**,*reft* commands) should also fall in this same temperature range. If not and **MP**,*alpx* was used, a note will be output. If not, and **MP**,*ctex* or **MP**,*thsx* was used, an error message will be output.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

MPTGEN, *STLOC*, *NUM*, *TSTRT*, *TINC*

Adds temperatures to the temperature table by generation.

PREP7: Materials

MP ME ST <> <> PR EM <> FL PP ED

STLOC
Starting location in table for generating temperatures. Defaults to last location filled + 1.

NUM
Number of temperatures to be generated (1-100).

TSTRT
Temperature assigned to *STLOC* location.

TINC
Increment previous temperature by *TINC* and assign to next location until all *NUM* locations are filled.

Notes

Adds temperatures to the temperature table by generation. May be used in combination (or in place of) the **MPTEMP** command.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Other>Change Mat Props>Generate Temp
Main Menu>Preprocessor>Material Props>Generate Temp
Main Menu>Solution>Other>Change Mat Props>Generate Temp

MPTRES, *Lab*, *MAT*

Restores a temperature table previously defined.

PREP7: Materials

MP ME ST <> <> PR EM <> FL PP ED

Lab
Material property label [**MP**].

MAT
Material reference number.

Notes

Restores into the database (from virtual space) a temperature table previously defined [**MP**] for a particular property. The existing temperature table in the database is erased before this operation.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Other>Change Mat Props>Restore Temps
Main Menu>Preprocessor>Material Props>Restore Temps
Main Menu>Solution>Other>Change Mat Props>Restore Temps

MPWRITE, *Fname*, *Ext*, --, LIB, *MAT*

Writes linear material properties in the database to a file (if the LIB option is not specified) or writes both linear and nonlinear material properties (if LIB is specified) from the database to a file.

PREP7: Materials

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify the LIB option, the default directory is the current working directory. If you specify LIB and you have specified a material library directory (via the **/MPLIB** command), that directory is the default. Otherwise, the default is the current working directory. If you use the default for your directory, you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

If you omit the LIB option, the default extension is MP. If you specify the LIB option, the default extension is units_MPL, where units is the system of units currently in use. (See the description of the **/UNITS** command.) For example, if **/UNITS** is set to BIN, the extension defaults to BIN_MPL.

- -

Unused field

LIB

The only value allowed for this field is the string "LIB."

The LIB option indicates that you wish to have properties associated with the material (MAT) written to the specified material library file using the material library file format. The material library file format is ASCII-text-based ANSYS command input. Certain commands associated with this format have been modified to interpret the string "_MATL" to mean the currently selected material. This feature makes the material library file independent of the material number in effect when the file was written; this enables you to restore the properties into the ANSYS database using the material number of your choice. The LIB option also enables you to save both linear and nonlinear properties. If you omit the LiB option, you can save linear properties only.

MAT

Specifies the material to be written to the named material library file. *There is no default; you must either specify a material or omit the MAT argument.* Even if you specify a *MAT* value, the ANSYS program ignores it if the LIB argument is not specified.

Notes

Writes linear material properties currently in the database to a file. The file is rewound before and after writing.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Export Library
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Import Library
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Select Units
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Write to File
Main Menu>Preprocessor>Material Props>Material Library>Export Library
Main Menu>Preprocessor>Material Props>Material Library>Import Library
Main Menu>Preprocessor>Material Props>Material Library>Select Units
Main Menu>Preprocessor>Material Props>Write to File
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Export Library
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Import Library
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Select Units
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Write to File

/MREP, *NAME*, *ARG1*, *ARG2*, *ARG3*, *ARG4*, *ARG5*, *ARG6*, *ARG7*, *ARG8*, *ARG9*, *ARG10*, *ARG11*, *ARG12*, *ARG13*, *ARG14*, *ARG15*, *ARG16*, *ARG17*, *ARG18*

Enables you to reissue the graphics command macro "name" during a replot or zoom operation.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

NAME

The name identifying the macro file or macro block on a macro library file. The name can contain up to eight characters maximum and must begin with a letter.

ARG1, *ARG2*, *ARG3*, *ARG4*, *ARG5*, *ARG6*, *ARG7*, *ARG8*, *ARG9*, *ARG10*, *ARG11*, *ARG12*, *ARG13*, *ARG14*, *ARG15*, *ARG16*, *ARG17*, *ARG18*

Values to be passed into the file or block.

Notes

This command reissues the graphics command macro "name" during a replot operation [**/REPLOT**] or a zoom [**ZOOM**] operation. The ANSYS program passes the command macro arguments to the replot and zoom feature for use by the graphics macro. You should place the **/MREP** command at the end of the graphics command macro, following the last graphics command within the macro, to enable the replot or zoom feature.

Menu Paths

This command cannot be accessed from a menu.

MSADV, *SPNUM*, *MTHA*

Specifies the approach to discretize the advection term in a species transport equation.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

SPNUM

Species number, from 1 to 6. Must be specified.

MTHA

Choice of approach to discretize the advection term:

MSU

Monotone Streamline Upwind (MSU) approach.

SUPG

Streamline Upwind / Petrov-Galerkin (SUPG) approach (default).

COLG

Collocated Galerkin (COLG) approach.

Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis. See the *ANSYS, Inc. Theory Reference* for more information on the advection term.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSAVE, *Key*

Sets the solver memory saving option. This option only applies to the PCG and DPCG solvers.

SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> <> PP ED

Key
Activation key:

0 or OFF
Use global assembly for the stiffness matrix of the entire model (default).

1 or ON
Use an element-by-element approach when possible to save memory during the solution. In this case, the global stiffness matrix is not assembled; element stiffness is regenerated during iterations.

Notes

The memory saving feature only applies to parts of the model using SOLID92, SOLID95, SOLID186, and/or SOLID187 elements with linear material properties. The following conditions must also be true:

- The PCG solver has been specified
- The analysis is either a static analysis or a modal analysis with the PowerDynamics method chosen
- Small strains (**NLGEOM**,OFF) (SOLID186, SOLID187 only)
- No stress stiffening effects (**SSTIF**) or prestress effects (**PSTRES**) included.
- All nodes on the supported element types must be defined (i.e., the midside nodes cannot be removed using the **EMID** command).
- For elements with thermally dependent material properties, **MSAVE**,ON only applies to elements with uniform temperatures prescribed.
- The default element coordinate system for SOLID92, SOLID95, SOLID186, or SOLID187 elements must be used.

Other parts of the model that don't meet the above criteria will be solved using global assembly. This command will result in memory savings of up to 70% over the global assembly approach for the part of the model that meets the criteria. Depending on the processor speed and manufacturer of the computer on which you are running ANSYS, the solution time may increase or decrease when this feature is used.

This memory saving feature runs in parallel when multiple processors are used with the **/CONFIG** command. The gain in performance with using multiple processors with this feature turned on should be similar to the default case when this feature is turned off. Performance also improves when using the 2 x 2 x 2 integration option for SOLID95 elements.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

MSCAP, *SPNUM*, *Capkey*, *UPPER*, *LOWER*

Activates and controls mass fraction capping for a species.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

SPNUM
Species number, from 1 to 6. Must be specified.

Capkey
Key to activate mass fraction capping:

OFF
Capping not enforced (default).

ON
Capping will be enforced.

UPPER, *LOWER*
Upper and lower bounds on mass fraction if capping is activated. Default to 1.0 and 0.0 respectively.

Command Default

No mass fraction capping.

Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSDATA, *ALGEB*, *UGAS*

Defines multiple species data applicable to all species.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

ALGEB
The algebraic species number whose mass fraction is calculated by subtracting the sum of the mass fractions of all other species from 1.0. This ensures that the sum of the mass fractions of all the species is 1.0. Defaults to 2.

UGAS
The universal gas constant. Defaults to 8314.3 (SI units).

Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSHAPE, *KEY*, *Dimension*

For elements that support multiple shapes, specifies the element shape to be used for meshing.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

KEY
Key indicating the element shape to be used:

0
Mesh with quadrilateral-shaped elements when *Dimension* = 2-D mesh with hexahedral-shaped elements when *Dimension* = 3-D.

1
Mesh with triangle-shaped elements when *Dimension* = 2-D mesh with tetrahedral-shaped elements when *Dimension* = 3-D.

Dimension
Specifies the dimension of the model to be meshed:

2D
2-D model (area mesh).

3D
3-D model (volume mesh).

Command Default

Since specification of element shape [**MSHAPE**] and meshing type [**MSHKEY**] are so closely related, the element shape that ANSYS meshes with depends on the combination of the values that are set for the two commands. The table below explains what happens when you fail to specify values for these settings.

Your action...	How it affects the mesh...
You issue the **MSHAPE** command with no arguments.	ANSYS uses quadrilateral-shaped or hexahedral-shaped elements to mesh the model, depending on whether you are meshing an area or a volume.
You do not specify an element shape, but you do specify the type of meshing to be used [**MSHKEY**].	ANSYS uses the default shape of the element to mesh the model. It uses the type of meshing that you specified.

Your action...	How it affects the mesh...
You specify neither an element shape nor the type of meshing to be used.	ANSYS uses the default shape of the element to mesh the model. It uses whichever type of meshing is the default for that shape.

Notes

If no value is specified for *Dimension* the value of *KEY* determines the element shape that will be used for both 2-D and 3-D meshing. In other words, if you specify **MSHAPE**,0, quadrilateral-shaped and hexahedral-shaped elements will be used. If you specify **MSHAPE**,1, triangle-shaped and tetrahedral-shaped elements will be used.

The **MSHAPE**, **MSHKEY**, and **MSHMID** commands replace the functionality that was provided by the **ESHAPE** command in ANSYS 5.3 and earlier releases.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>4 to 6 sided
Main Menu>Preprocessor>Meshing>Mesher Opts

MSHCOPY, *KEYLA*, *LAPTRN*, *LACOPY*, *KCN*, *DX*, *DY*, *DZ*, *TOL*, *LOW*, *HIGH*

Simplifies the generation of meshes that have matching node element patterns on two different line groups (in 2-D) or area groups (3-D).

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

KEYLA
Copy line mesh (default) if LINE, 0 or 1. Copy area mesh if AREA, or 2.

LAPTRN
Meshed line/area to be copied, or a component name containing a list. If *LAPTRN* = P, graphical picking is enabled (valid only in the GUI).

LACOPY
Unmeshed line/area to get copied mesh, or a component name containing a list. If *LACOPY* = P, graphical picking is enabled (valid only in the GUI).

KCN
In coordinate system *KCN*, *LAPTRN* + *DX DY DZ* = *LACOPY*.

DX, *DY*, *DZ*
Node location increments in the active coordinate system (DR, Dθ, DZ for cylindrical, DR, Dθ, DΦ for spherical or toroidal).

TOL
Tolerance. Defaults to 1.e--4.

LOW
Name of low node component to be defined (optional).

HIGH
Name of high node component to be defined (optional).

Notes

Matching meshes are used for rotational (cyclic) symmetry, or for contact analysis using coupling or node-to-node gap elements. See Using CPCYC and MSHCOPY Commands in the *ANSYS Modeling and Meshing Guide* for more information.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Area Mesh
Main Menu>Preprocessor>Modeling>Copy>Line Mesh

MSHKEY, *KEY*

Specifies whether free meshing or mapped meshing should be used to mesh a model.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

KEY

Key indicating the type of meshing to be used:

0
Use free meshing (the default).

1
Use mapped meshing.

2
Use mapped meshing if possible; otherwise, use free meshing. If you specify **MSHKEY**,2, SmartSizing will be inactive even while free meshing non-map-meshable areas.

Command Default

As stated above, free meshing is the default. However, since the **MSHKEY** and **MSHAPE** settings are closely related, you should refer to the table in the **MSHAPE** command description for more information about defaults.

Notes

The **MSHKEY**, **MSHAPE**, and **MSHMID** commands replace the functionality that was provided by the **ESHAPE** command in ANSYS 5.3 and earlier releases.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>3 or 4 sided
Main Menu>Preprocessor>Meshing>Mesh>Areas>Target Surf
Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>4 to 6 sided
Main Menu>Preprocessor>Meshing>Mesher Opts

MSHMID, *KEY*

Specifies placement of midside nodes.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

KEY

Key indicating placement of midside nodes:

0

Midside nodes (if any) of elements on a region boundary follow the curvature of the boundary line or area (the default).

1

Place midside nodes of all elements so that element edges are straight. Allows coarse mesh along curves.

2

Do not create midside nodes (elements will have removed midside nodes).

Notes

The **MSHMID**, **MSHAPE**, and **MSHKEY** commands replace the functionality that was provided by the **ESHAPE** command in ANSYS 5.3 and earlier releases.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesher Opts

MSHPATTERN, *KEY*

Specifies pattern to be used for mapped triangle meshing.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

KEY

Key indicating triangle pattern to be used (the figures below illustrate the pattern that will be used for each value of *KEY*):

0

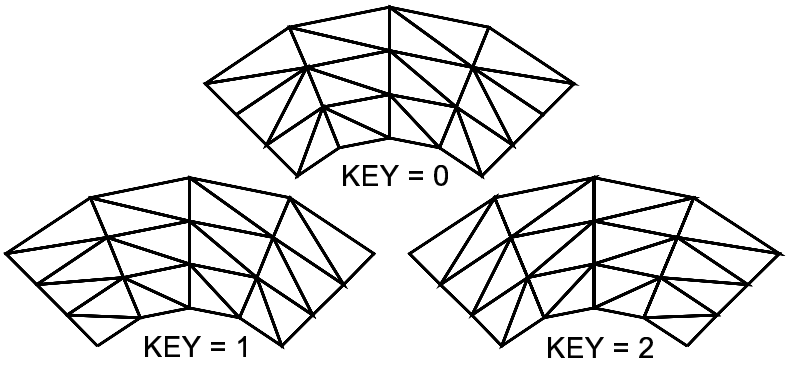
Let ANSYS choose the pattern (the default). ANSYS maximizes the minimum angle of the triangular-shaped elements that are created.

1

Unidirectional split at node I.

2

Unidirectional split at node J.



Notes

"Mapped triangle meshing" refers to the ANSYS program's ability to take a map-meshable area and mesh it with triangular elements, based on the value of **MSHPATTERN**,*KEY*. This type of meshing is particularly useful for analyses that involve the meshing of rigid contact elements.

The **MSHPATTERN** command is valid only when you have specified that ANSYS use triangle-shaped elements [**MSHAPE**,1,2D] (or you are meshing with an element that supports only triangles), and you have also specified mapped meshing [**MSHKEY**,1] to mesh an area.

For details about mapped meshing with triangles, see the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesher Opts

MSMASS, SPNUM, *Value*

Specifies the mass type for a transient species analysis.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

SPNUM
Species number, from 1 to 6. Must be specified. If ALL, all numbers are used.

Value
Mass type for transient species analysis:

LUMP
Lumped mass matrix (default).

CONS
Consistent mass matrix.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSMETH, *SPNUM*, *KEY*

Specifies the method of solution of the species transport equations.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

SPNUM
Species number, from 1 to 6. Must be specified.

KEY
Key defining the method of solution for the specified species number:

0
No solution of equations for species *SPNUM*

1
Tri-Diagonal Matrix Algorithm (TDMA).

2
Conjugate residual method.

3
Preconditioned Conjugate Residual method.

4
Preconditioned Generalized Minimum Residual (PGMR) solution method (default).

5
Sparse Direct method.

6
Preconditioned BiCGStab method (PBCGM).

Command Default

The Preconditioned Generalized Minimum Residual (PGMR) method is used for all species.

Notes

The Tri-Diagonal Matrix Algorithm (TDMA) method is a special version of the standard Gauss-Seidel iterative method for the solution of sets of algebraic equations. The number of iterations (sweeps) to be performed is specified with the **MSSOLU** command. No convergence criterion is required for the TDMA method.

Three methods are semi-direct solution methods based on search directions. They are conjugate direction iterative techniques which develop a solution as a linear combination of search directions. The Conjugate Residual method requires the least memory, but stalls when solving ill-conditioned problems. The Preconditioned Conjugate Residual method requires much more memory but performs better for ill-conditioned matrix problems. The PGMR method is memory-intensive; by necessity, it incorporates a tight convergence criterion. The Preconditioned BiCGStab method (PBCGM) requires less memory than the PGMR method and sometimes performs better than the PGMR method. The number of search vectors and the convergence criterion are specified with the **MSSOLU** command.

The Sparse Direct method is based on Gaussian elimination to factorize the matrix. This method is memory intensive and creates temporary files on the hard disk. It is robust and can be used for symmetric as well as nonsymmetric equation systems.

See the *ANSYS Fluids Analysis Guide* for additional information on the FLOTRAN Solvers.

This command is valid for the multiple species transport option in a FLOTRAN analysis.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSMIR, *SPNUM*, *Value*

Sets modified inertial relaxation factors for multiple species.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

SPNUM
Species number, from 1 to 6. Must be specified.

Value
Modified inertial relaxation factor. *Value* defaults to 0 (modified inertial relaxation off).

Notes

Value must be a positive real number. A *Value* between 0.1 and 1.0 is recommended. A larger *Value* provides a more robust scheme, but it may yield a slower convergence.

This command is valid for the multiple species transport option in a FLOTRAN analysis.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSNOMF, *SPNUM*, *FRACTION*

Specifies the initial value of nominal mass fraction for a species.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

SPNUM
Species number, from 1 to 6. Must be specified.

FRACTION
The initial mass fraction of the entire problem domain for this species. Defaults to 0.0. The sum of the mass fractions for all species should equal 1.0.

Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis and is required if the CMIX option has been activated for a property [**FLDATA7**,PROT,*Label*,CMIX].

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSPROP, *SPNUM*, *Label*, *Type*, *NOMINAL*, *COF1*, *COF2*, *COF3*

Defines the fluid properties of a species.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

SPNUM
Species number, from 1 to 6. Must be specified.

Label
Label identifying the property being defined:

DENS
Density.

VISC
Viscosity.

COND
Thermal conductivity.

MDIF
Mass diffusion coefficient.

SPHT
Specific heat.

Type
Type of property:

CONSTANT
Constant property (default). The property does not vary with temperature.

LIQUID
Liquid property. Density varies according to a second order polynomial relationship, and all other properties follow Sutherland's law for liquids.

GAS
Gas property. Density varies according to the ideal gas law, and all other properties follow Sutherland's law for gases.

NOMINAL
Nominal value of the property being defined. For CONSTANT fluid types, the property remains at this value and does not vary. For GAS and LIQUID property types, this is the value of the property corresponding to the temperature defined by *COF1*.

COF1
Temperature corresponding to the *NOMINAL* value of the property (for GAS and LIQUID property types only; see Notes section). Not required for *Label* = SPHT.

COF2, *COF3*
Second and third coefficients for temperature variation of the property. Not required for *Label* = SPHT.

Notes

If the property type is CONSTANT, the equation used is as follows:

Lab = *NOMINAL*

If the property type is LIQUID, a second order polynomial relationship is used for density, and Sutherland's law for liquids is used for the other properties:

DENS = *NOMINAL* + *COF2**(T-*COF1*) + *COF3**(T-*COF1*)^2

Property = *NOMINAL* * EXP[*COF2**(1/T-1/*COF1*) + *COF3**(1/T-1*COF1*)^2]

where T is the temperature of the node where the property is being calculated.

If the property type is GAS, the ideal gas law is used for density, and Sutherland's law for gases is used for other properties:

DENS = *NOMINAL* * (P/*COF2* / (T/*COF1*)

Property = *NOMINAL* * (T/*COF1*)$^{1.5}$ * (*COF1* + *COF2*)/(T + *COF2*)]

where P and T are the pressure and temperature of the node where the property is being calculated.

Specific heat is always a CONSTANT. Also, property types (*Type*) such as TABLE, USER, POWL, BIN, etc. are not available for individual species. They are valid only for the bulk fluid.

If a property type is entered incorrectly (e.g. a misspelling), a CONSTANT property type is assumed and the property is assigned the NOMINAL value.

This command is valid for the multiple species transport option in a FLOTRAN analysis.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSQUAD, *QDIF*, *QSRC*

Specifies the quadrature order for multiple species elements.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

QDIF
Quadrature order for diffusion term integration:

0
One-point quadrature (default).

1
Same as 0, except a distributed value of temperature is used to evaluate temperature-dependent properties.

2
Two-point quadrature (default for axisymmetric models).

QSRC
Quadrature order for source term integration:

0
One-point quadrature (default).

1
Same as 0, except a distributed value of temperature is used to evaluate temperature-dependent properties.

2
Two-point quadrature (default for axisymmetric models).

Command Default

As described above.

Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSRELAX, *SPNUM*, *CONC*, *MDIF*, *EMDI*, *STAB*

Specifies relaxation factors for a multiple species transport analysis.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

SPNUM
Species number, from 1 to 6. Must be specified.

CONC
Species concentration relaxation factor. Defaults to 0.5.

MDIF
Mass diffusion coefficient relaxation factor. Defaults to 0.5.

EMDI
Effective mass diffusion coefficient relaxation factor (used for turbulent flow). Defaults to 0.5.

STAB
Inertial relaxation factor for solution of the transport equation. Defaults to $1.0 \times 10^{+20}$.

Command Default

As described above for each relaxation factor.

Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSSOLU, *SPNUM*, *NSWEEP*, *MAXI*, *NSRCH*, *CONV*, *DELMAX*

Specifies solution options for multiple species transport.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> FL PP ED

SPNUM
Species number, from 1 to 6. Must be specified.

NSWEEP
Number of Tri-Diagonal Matrix Algorithm (TDMA) sweeps. Valid only for the TDMA method [**MSMETH**]. Defaults to 100.

MAXI
Maximum number of iterations allowed for the semi-direct methods (conjugate residual and preconditioned conjugate residual methods, chosen with the **MSMETH** command). Defaults to 1000.

NSRCH
Number of search vectors used for the semi-direct methods. Defaults to 2. If you are using the Preconditioned BiCGStab Method (PBCGM) solver, the number of search directions is 1 to 8, with 2 as the default. New search vectors are made orthogonal to *NSRCH* previous vectors in the solution of the unsymmetric matrix systems.

CONV
Convergence criterion for the semi-direct methods. It represents the factor by which the inner product of the residual vector is reduced during the solution of the equations at any global iteration. Defaults to 1.0×10^{-12}. If the convergence criterion has not been achieved, the algebraic solver issues a warning message, and the execution of FLOTRAN continues normally.

DELMAX
Minimum normalized rate of change which will permit the semi-direct solution methods to continue. Used to terminate the semi-direct solvers in the event that stall occurs. Defaults to 1.0×10^{-9}. If the methods stall, the solver increments the solution only a very small amount despite the fact that the correct solution has been not been achieved (or perhaps even approached). The maximum nodal difference between the solutions, normalized to the value of the variable, is compared to *DELMAX*, and the solution is terminated if the value is less than *DELMAX*. Termination of the algebraic solver due to the small rate of change is considered a normal function, and no warning message is printed. Execution of FLOTRAN continues normally.

Command Default

As described above.

Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSSPEC, *SPNUM*, *Name*, *MOLWT*, *SCHMIDT*

Specifies the name, molecular weight, and Schmidt number of a species.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

SPNUM
Species number, from 1 to 6. Must be specified.

Name
Name to be assigned to the species, up to 4 characters long. Defaults to SP01 for species 1, SP02 for species 2, ..., SP06 for species 6. This name can be used in place of the species number when specifying mass fraction boundary conditions and in postprocessing.

Note — The GUI always shows the default names, not the user-defined names.) *Name* should not be the same as an existing degree of freedom label.

MOLWT
Molecular weight for the species. Required only for gases (determined by the property type on **MSPROP** command). Defaults to 29.0.

SCHMIDT
Schmidt number (diffusion term divisor) for the species. Required only for gases (determined by the property type on **MSPROP** command). Defaults to 1.0.

Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

/MSTART, *Label*, *KEY*

Controls the initial GUI components.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Label

Label identifying the GUI component:

ZOOM
Pan, Zoom, Rotate dialog box, off by default.

WORK
Offset Working Plane dialog box, off by default.

WPSET
Working Plane Settings dialog box, off by default.

ABBR
Edit Toolbar/Abbreviations dialog box, off by default.

PARM
Scalar Parameters dialog box, off by default.

SELE
Select Entities dialog box, off by default.

ANNO
Annotation dialog box, off by default.

HARD
Hard Copy dialog box, off by default.

UTIL
Turns on the pre-ANSYS 6.1 (UIDL) GUI, off by default.

KEY

Switch value:

OFF or 0
Component does not appear when GUI is initialized.

ON or 1
Component appears when GUI is initialized.

Command Default

Same as *Label* defaults.

Notes

Controls which components appear when the Graphical User Interface (GUI) is initially brought up. This command is valid only *before* the GUI is brought up [**/MENU**,ON] and is intended to be used in the start81.ans file. It only affects how the GUI is *initialized*; you can always bring up or close any component once you are in the GUI.

This command is valid only at the Begin Level.

Menu Paths

This command cannot be accessed from a menu.

MSTERM, *SPNUM*, *STER*, *TTER*

Sets the convergence monitors for species.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

SPNUM
Species number, from 1 to 6. Must be specified.

STER
Termination criteria for steady-state analysis. Defaults to 1×10^{-8}.

TTER
Termination criteria for transient analysis. Defaults to 1×10^{-6}.

Notes

Repeat command to set each species number as required.

All specified criteria must be met before the case is terminated.

If a termination criterion for a specific species number is set negative, the termination check is ignored for that particular species.

This command is valid for the multiple species transport option in a FLOTRAN analysis.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MSVARY, *SPNUM*, *Lab*, *Key*

Allows species properties to vary between global iterations.

PREP7: FLOTRAN Multiple Species

MP <> <> <> <> <> <> <> FL PP ED

SPNUM
Species number, from 1 to 6. Must be specified.

Lab
Label identifying the species property:

DENS
Density.

VISC
Viscosity.

COND
Thermal conductivity.

MDIF
Mass diffusion coefficient.

Key
Key to allow property variation between global iterations:

OFF
Variation not allowed (default).

ON
Variation allowed.

Command Default

No property is allowed to vary between global iterations within a load step.

Notes

This command is valid for the multiple species transport option in a FLOTRAN analysis.

Menu Paths

Main Menu>Preprocessor>FLOTRAN Set Up>Multiple Species
Main Menu>Solution>FLOTRAN Set Up>Multiple Species

MXPAND, *NMODE*, *FREQB*, *FREQE*, *Elcalc*, *SIGNIF*

Specifies the number of modes to expand and write for a modal or buckling analysis.

SOLUTION: Dynamic Options
SOLUTION: Nonlinear Options

MP ME ST <> <> PR <> <> <> PP ED

NMODE
Number of modes to expand and write. If blank, expand and write all modes within the frequency range specified.

FREQB
Beginning, or lower end, of frequency range of interest. If *FREQB* and *FREQE* are both blank, expand and write the number of modes specified without regard to the frequency range. Defaults to the entire range.

FREQE
Ending, or upper end, of frequency range of interest.

Elcalc
Element calculation key:

NO
Do not calculate element results and reaction forces (default).

YES
Calculate element results and reaction forces, as well as the nodal degree of freedom solution.

SIGNIF
Expand only those modes whose significance level exceeds the *SIGNIF* threshold. The significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not expanded. The higher the *SIGNIF* threshold, the fewer the number of modes expanded. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. *SIGNIF* value is only used for single-point or DDAM response (**SPOPT**,SPRS or DDAM) analyses.

Command Default

Mode shapes (displacements) are expanded and output to the results file (so that if you need to inspect the mode shapes, it is not necessary to run the analysis again). Element stresses are not expanded.

Notes

Specifies the number of modes to expand and write over a frequency range for a modal (**ANTYPE**,MODAL) or buckling (**ANTYPE**,BUCKLE) analysis. For reduced analyses, an expansion is required. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Expand Modes
Main Menu>Solution>Analysis Type>Analysis Options
Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Expand Modes

N Commands

N, *NODE*, *X*, *Y*, *Z*, *THXY*, *THYZ*, *THZX*
Defines a node.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

NODE
Node number to be assigned. A previously defined node of the same number will be redefined. Defaults to the maximum node number used +1.

X, *Y*, *Z*
Node location in the active coordinate system (R, θ, Z for cylindrical, R, θ, Φ for spherical or toroidal). If *X* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

THXY
First rotation about nodal Z (positive X toward Y).

THYZ
Second rotation about nodal X (positive Y toward Z).

THZX
Third rotation about nodal Y (positive Z toward X).

Notes

Defines a node in the active coordinate system [**CSYS**]. The nodal coordinate system is parallel to the global Cartesian system unless rotated. Rotation angles are in degrees and redefine any previous rotation angles. See the **NMODIF**, **NANG**, and **NROTAT** commands for other rotation options.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>In Active CS
Main Menu>Preprocessor>Modeling>Create>Nodes>On Working Plane

---

NANG, *NODE*, *X1*, *X2*, *X3*, *Y1*, *Y2*, *Y3*, *Z1*, *Z2*, *Z3*
Rotates a nodal coordinate system by direction cosines.

PREP7: Nodes

MP ME ST DY <> PR EM <> <> PP ED

NODE
Rotate coordinate system of this node.

X1, *X2*, *X3*
Global X, Y, Z components of a unit vector in new nodal X direction.

Y1, *Y2*, *Y3*
Global X, Y, Z components of a unit vector in new nodal Y direction.

Z1, *Z2*, *Z3*
Global X, Y, Z components of a unit vector in new nodal Z direction.

Notes

Rotates a nodal coordinate system to the orientation specified by the X, Y and Z direction cosines. Existing rotation specifications on the node are redefined. If only two of the three unit vectors are specified, the third is defined according to the right hand rule. It is the responsibility of the user to ensure that input direction cosines are orthogonal in a right-handed system.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>RotateNode>By Vectors
Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>By Vectors

---

NCNV, *KSTOP*, *DLIM*, *ITLIM*, *ETLIM*, *CPLIM*

Sets the key to terminate an analysis.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR EM <> <> PP ED

KSTOP
Program behavior upon nonconvergence:

0
Do not terminate the analysis if the solution fails to converge.

1
Terminate the analysis and the program execution if the solution fails to converge (default).

2
Terminate the analysis, but not the program execution, if the solution fails to converge.

DLIM
Terminates program execution if the largest nodal DOF solution value (displacement, temperature, etc.) exceeds this limit. Defaults to 1.0E6 for all DOF except MAG and A and FLOTRAN DOF. Defaults to 1.0E10 for MAG and A. Defaults to 1.0E20 for the FLOTRAN VX, VY, VZ, PRES, and TEMP DOF. Does not apply to the FLOTRAN ENKE and ENDS DOF.

ITLIM
Terminates program execution if the cumulative iteration number exceeds this limit (defaults to infinity).

ETLIM
Terminates program execution if the elapsed time (seconds) exceeds this limit (defaults to infinity).

CPLIM
Terminates program execution if the CPU time (seconds) exceeds this limit (defaults to infinity).

Command Default

As described for each argument above.

Notes

Sets the key to terminate an analysis if not converged, or if any of the following limits are exceeded for nonlinear and full transient analyses: DOF (displacement), cumulative iteration, elapsed time, or CPU time limit. Applies

only to static and transient analyses (**ANTYPE**,STATIC and **ANTYPE**,TRANS), and FLOTRAN analyses. Time limit checks are made at the end of each equilibrium iteration. This command is ignored during an optimization run.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Advanced NL
Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Criteria to Stop
Main Menu>Solution>Analysis Type>Sol'n Controls>Advanced NL
Main Menu>Solution>Load Step Opts>Nonlinear>Criteria to Stop

NDELE, *NODE1*, *NODE2*, *NINC*

Deletes nodes.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

NODE1, *NODE2*, *NINC*

Delete nodes from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and all selected nodes [**NSEL**] are deleted. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1*.

Notes

Deletes selected nodes that are not connected to elements. Nodes may also be redefined instead of deleted, if desired. Boundary conditions (displacements, forces, etc.) as well as any coupling or constraint equations containing the deleted nodes are also deleted.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements
Main Menu>Preprocessor>Modeling>Delete>Nodes

NDIST, *ND1*, *ND2*

Calculates and lists the distance between two nodes.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

ND1

First node in distance calculation. If *ND1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

ND2

Second node in distance calculation.

Notes

NDIST lists the distance between nodes *ND1* and *ND2*, as well as the current coordinate system offsets from *ND1* to *ND2*, where the X, Y, and Z locations of *ND1* are subtracted from the X, Y, and Z locations of *ND2* (respectively) to determine the offsets. **NDIST** is valid in any coordinate system except toroidal [**CSYS**,3].

NDIST returns a variable, called "*_RETURN*," which contains the distance value. You can use this value for various purposes, such as the calculation of distributed loads. In interactive mode, you can access this command by using the Model Query Picker (**Utility Menu> List> Picked Entities**), where you can also access automatic annotation functions and display the value on your model.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>ND distances

NDSURF, *Snode*, *Telem*, *DIMN*

Generates surface elements overlaid on the edge of existing elements and assigns the extra node as the closest fluid element node.

PREP7: Elements

MP ME <> <> <> PR <> <> <> <> ED

Snode
Component name for the surface nodes of the solid elements.

Telem
Component name for the target fluid elements.

DIMN
Model dimensionality:

2
2-D model.

3
3-D model.

Notes

This command macro is used to generate surface effect elements (SURF151 or SURF152) overlaid on the surface of existing plane or solid elements and, based on proximity, to determine and assign the extra node (FLUID116) for each surface element.

The surface nodes of the plane or solid elements must be grouped into a node component and the fluid elements must be grouped into an element component and named using the **CM** command. The names must be enclosed in single quotes (e.g., 'NOD') when the **NDSURF** command is manually typed in.

When using the GUI method, node and element components are created through the picking dialog boxes associated with this command.

The macro is applicable for the SURF151, SURF152, and FLUID116 element types.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Area to Fluid
Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Line to Fluid
Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Node to Fluid

---

NEQIT, *NEQIT*

Specifies the maximum number of equilibrium iterations for nonlinear analyses.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR EM <> <> PP ED

NEQIT
Maximum number of equilibrium iterations allowed each substep.

Command Default

When **SOLCONTROL**,ON, *NEQIT* is set between 15 and 26 depending on the physics of the problem. When **SOLCONTROL**,OFF, *NEQIT* defaults to 25 for all cases.

Notes

See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL**,ON and **SOLCONTROL**,OFF.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Equilibrium Iter
Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Solution>Load Step Opts>Nonlinear>Equilibrium Iter

/NERR, *NMERR*, *NMABT*, --, *IFKEY*, *NUM*

Limits the number of warning and error messages displayed.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

NMERR
Maximum number of warning and error messages displayed per command. Defaults to 5 for interactive runs with the GUI turned on, 20 for interactive runs with the GUI turned off, 200 for batch runs. If *NMERR* is negative,

the absolute value of *NMERR* is used as the maximum number of warning and error messages written to the error file (**file.ERR**) per command, as well as the maximum number of messages displayed per command.

NMABT
Maximum number of warning and error messages allowed per command before run aborts (must be greater than zero). Maximum value is 99,999,999. Defaults to 10,000.

--
Unused field.

IFKEY
Specifies whether or not to abort if an error occurs during a **/INPUT** operation:

0*or*OFF
Do not abort. This option is the default.

1*or*ON
Abort.

NUM
The number of invalid command warnings before a stop warning will be issued:

0
Disables the stop warning/error function.

n
An integer value representing the number of warnings that will be encountered before prompting the user to stop (default = 5). The first error encountered will ALWAYS result in a prompt.

Note — Invalid command warnings and error tracking are mutually exclusive.

Command Default

As described above.

Notes

Limits the number of warning and error messages displayed for any one command in an interactive run.

Warning and error messages continue to be written to **Jobname.ERR** regardless of these limits (unless *NMERR* is negative).

Issue this command with *NUM* = *n* to specify the number of "invalid command" warnings to be encountered before the user is prompted to stop. You can then continue or abort the run. If you choose to abort the run, the log file can be saved so that any of the processing up to that point can be appended to an input that rectifies the condition. A batch run always aborts on the first error. Issue **/NERR**,STAT to list current settings.

Issue **/NERR**,DEFA to reset values to initial defaults.

An *IFKEY* value of 1 or ON causes the ANSYS program to abort immediately upon encountering an error during a file **/INPUT** operation. However, use of this option may cause the following conditions to occur:

- The **/INPUT** command may abort if issued for a log file (***jobname*.log**).
- Some macros may abort.

- ANSYS Connection may fail after reading only a small portion of a CAD model.

The command is valid in any processor.

Menu Paths

Utility Menu>MenuCtrls>Message Controls

NFORCE, *ITEM*

Sums the nodal forces and moments of elements attached to nodes.

POST1: Special Purpose

MP ME ST <> <> PR EM <> <> PP ED

ITEM
Specifies the selected set of nodes for summing forces and moments for contact elements.

blank
Sums the nodal forces of elements for all selected nodes and excludes contact elements (elements 169-174).

CONT
Sums the nodal forces of elements for contact nodes only.

BOTH
Sums the nodal forces of elements for all selected nodes, including contact elements.

Notes

Sums and prints, in each component direction for each selected node, the nodal force and moment contributions of the selected elements attached to the node. If all elements are selected, the sums are usually zero except where constraints or loads are applied. The nodal forces and moments may be displayed [**/PBC**,FORC and **/PBC**,MOME]. Use **PRESOL** to print nodal forces and moments on an element-by-element basis. You can use the **FORCE** command to specify which component (static, damping, inertia, or total) of the nodal load is to be used. Nodal forces associated with surface loads are not included.

This vector sum is printed in the global Cartesian system. Moment summations are about the global origin unless another point is specified with the **SPOINT** command. The summations for each node are printed in the global Cartesian system unless transformed [**RSYS**]. This command is generally not applicable to axisymmetric models because moment information from the **NFORCE** command is not correct for axisymmetric elements.

Selecting a subset of elements [**ESEL**] and then issuing this command will give the forces and moments required to maintain equilibrium of that set of elements. The effects of nodal coupling and constraint equations are ignored. The option *ITEM*=CONT provides the forces and moments for the contact elements (surface-to-surface only for CONTA171, CONTA172, CONTA173, and CONTA174). Setting *ITEM* = BOTH provides the forces and moments for all selected nodes, including contact elements.

This command also includes the **FSUM** command function which vectorially sums and prints, in each component direction for the total selected node set, the nodal force and moment contributions of the selected elements attached to the selected node set.

Menu Paths

Main Menu>General Postproc>Nodal Calcs>Sum @ Each Node

NGEN, *ITIME*, *INC*, *NODE1*, *NODE2*, *NINC*, *DX*, *DY*, *DZ*, *SPACE*

Generates additional nodes from a pattern of nodes.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

ITIME, *INC*

Do this generation operation a total of *ITIME* times, incrementing all nodes in the given pattern by *INC* each time after the first. *ITIME* must be > 1 for generation to occur.

NODE1, *NODE2*, *NINC*

Generate nodes from the pattern of nodes beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and the pattern is all selected nodes [**NSEL**]. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

DX, *DY*, *DZ*

Node location increments in the active coordinate system (DR, Dθ, DZ for cylindrical, DR, Dθ, DΦ for spherical or toroidal).

SPACE

Spacing ratio. Ratio of last division size to first division size. If > 1.0, divisions increase. If < 1.0, divisions decrease. Ratio defaults to 1.0 (uniform spacing).

> *Note* — The *average* spacing ratio remains 1.0, such that the location of the last generated set will be the same regardless of *SPACE*. *SPACE* only serves to skew the position of the nodes between the pattern set and the last set.

Notes

Generates additional nodes from a given node pattern. Generation is done in the active coordinate system. Nodes in the pattern may have been generated in any coordinate system.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Nodes>Copy

NKPT, *NODE*, *NPT*

Defines a node at an existing keypoint location.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

NODE

Arbitrary reference number for node. If zero or blank, defaults to the highest node number +1 [**NUMSTR**].

NPT
Keypoint number defining global X, Y, Z location. If *NPT* = All, then a node will be placed at each selected keypoint. If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>On Keypoint

NLDIAG, *Label*, *Key*

Sets nonlinear diagnostics functionality.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR <> <> <> PP ED

Label
Specifies a diagnostic function:

NRRE
Store the Newton-Raphson residuals information.

EFLG
Identify or display elements or nodes that violate the criteria.

MAXF
Set a maximum number of files (*MaxFile*) to create. Information is written to **Jobname.nd*xxx*** or **Jobname.nr*xxx***, where *xxx* iterates from 001 through *MaxFile*. When the maximum number of files is reached, the counter resets to 001 and earlier files are overwritten.

Key
Sets the characteristics of the diagnostic function:

OFF, or 0
Suppress writing of diagnostic information. This value is the default.

ON, or 1
Writes diagnostic information to the **Jobname.nd*xxx*** or **Jobname.nr*xxx*** file.

STAT
Lists information about the diagnostic files in the current working directory.

DEL
Deletes all diagnostic files in the current working directory.

MaxFile
Sets the maximum number of files to create. Valid values are 1 through 999. The default is 10. Valid only when *Label* = MAXF.

Command Default

NLDIAG,NRRE,OFF

NLDIAG,EFLG,OFF

NLDIAG,MAXF,10 If you change the default *MaxFile* value, that value is retained until you change it again.

Notes

The **NLDIAG** command is a nonlinear diagnostics tool valid for nonlinear structural analyses. It is a debugging tool for use when you must restart after an unconverged solution. The command creates **Jobname.nd*xxx*** or **Jobname.nr*xxx*** files in the working directory to store the information you specify.

Newton-Raphson residual Issue a **NLDIAG**,NRRE,ON command to create **Jobname.nr*xxx*** files which store the relevant Newton-Raphson residual information for the last *MaxFile* equilibrium iterations. Issue a **NLDPOST**,NRRE,STAT command to list the load step, substep, time, and equilibrium iteration corresponding to each of the **Jobname.nr*xxx*** files in the working directory, then issue a **PLNSOL**,NRRES,,,,*FileID* command to point to the file from which you want to create a contour plot of your Newton-Raphson residuals.

Element components that violate criteria Issue a **NLDIAG**,EFLG,ON command to create **Jobname.nd*xxx*** files which store IDs for elements violating the following criteria:

- Too large a distortion
- Elements contain nodes that have near zero pivots for nonlinear analyses
- Too large a plastic/creep strain increment (**CUTCONTROL**)
- Elements for which mixed u-P constraints are not satisfied (mixed U-P option of 18*x* solid elements only)

For **NLDIAG**,EFLG,ON, all **Jobname.nd*xxx*** diagnostic files in the current (working) directory are deleted when you issue a new **SOLVE** command (or restart).

In the solution processor (**/SOLU**), use the STAT option to list the active status of this command. In the postprocessor (**/POST1**), issue a **NLDPOST**,EFLG,STAT command to list the load step, substep, time, and equilibrium iteration corresponding to each of the **Jobname.nd*xxx*** files in the working directory, then issue a **NLDPOST**,EFLG,CM,*FileID* command to create element components that violate the criteria.

For more information, see Section 8.10.2.1: Performing Nonlinear Diagnostics.

Menu Paths

Main Menu>Solution>Diagnostics>Nonlinear Diagnostics

NLDPOST, *Label*, *Key*, *FileID*, *Prefix*

Gets element component information from nonlinear diagnostic files.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR <> <> <> PP ED

Label

Specifies the type of command operation:

EFLG
: Element flag for nonlinear diagnostics.

NRRE
: Newton-Raphson residuals.

Key
Specifies the command action:

STAT
List information about the diagnostic files (**Jobname.nd*xxx*** or **Jobname.nr*xxx***) in the current directory.

For *Label* = EFLG, the listing gives a summary that associates the loadstep, substep, time, equilibrium iteration number, cumulative iteration number, and the number of elements that fail each criteria with a specific file ID (**Jobname.nd*xxx***). Use the list to create element components (via the the CM option) based on the cumulative iteration number.

For *Label* = NRRE, the listing provides a summary that associates the loadstep, substep, time, equilibrium iteration number, and cumulative iteration number with a specific file ID (**Jobname.nr*xxx***). Use the list to identify the respective file ID for creating Newton-Raphson residual contour plots (**PLNSOL**,NRRE,...,*FileID*).

DEL
Delete **Jobname.nd*xxx*** or **Jobname.nr*xxx*** files in the working directory, if any exist.

CM
Create component files for elements that violate criteria. This value is valid only when *Label* = EFLG.

FileID
Valid only when *Label* = EFLG and *Key* = CM, this value specifies file IDs:

IDnum
The file ID number. Creates the element components from the diagnostic files corresponding to the specified file ID number in the working directory.

ALL
Creates element components from all available diagnostic files residing in the working directory. This value is the default if you do not specify an *IDnum* value.

Prefix
Sets the prefix name for components. Specify up to 21 alphanumeric characters.

Command Default

The **NLDPOST** command has no defaults.

Notes

Based on the nonlinear diagnostic results (created via the **NLDIAG**,EFLG command), the **NLDPOST** command creates element components with predefined names.

The following table lists the diagnostic criteria and filenames (with specified prefix and without). Here *xxx* corresponds to the file ID (*FileID*) of **Jobname.nd*xxx*** or **Jobname.nr*xxx***.

Elements with:	If prefix is specified:	Without prefix specified:
Too large a plastic strain increment	*prefix_xxx*_eppl	nd*xxx*_eppl
Too large a creep strain increment	*prefix_xxx*_epcr	nd*xxx*_epcr
Unsatisfied mixed u-P constraints	*prefix_xxx*_mxup	nd*xxx*_mxup
Too much distortion	*prefix_xxx*_hdst	nd*xxx*_hdst

Elements with:	If prefix is specified:	Without prefix specified:
Negative/small pivots	*prefix_xxx*_pivt	nd*xxx*_pivt

For more information, see Section 8.10.2.1: Performing Nonlinear Diagnostics.

Menu Paths

Main Menu>General Postproc>Nonlinear Diagnostics

NLGEOM, *Key*

Includes large-deflection effects in a static or full transient analysis.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR <> <> <> PP ED

Key

Large-deflection key:

OFF
: Ignores large-deflection effects (that is, a small-deflection analysis is specified). This option is the default.

ON
: Includes large-deflection (large rotation) effects or large strain effects, according to the element type.

Command Default

Large-deflection effects are ignored.

Notes

Large-deflection effects are categorized as either large deflection (or large rotation) or large strain, depending on the element type. These are listed (if available) under Special Features in the input data table for each element in the *ANSYS Elements Reference*. If used during the solution (**/SOLU**), this command is valid only within the first load step.

In a large-deflection analysis, pressure loads behave differently than other load types. For more information, see Section 8.1.2.3: Load Direction in a Large-Deflection Analysis.

This command is also valid in PREP7.

Product Restrictions:

In ANSYS Professional, large deflection effects should not be turned on if 2-D solid (PLANE*n*) or 3-D solid (SOLID*n*) elements are defined.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic
Main Menu>Solution>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Sol'n Controls>Basic

NLHIST, *Key*, *Name*, *Item*, *Comp*, *NODE*, *ELEM*, *SHELL*, *LAYER*

Specify result items to track during solution.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR <> <> <> PP ED

Key

Specifies the command operation:

NSOL
: Nodal solution data.

ESOL
: Element nodal data.

PAIR
: Contact data (pair-based).

STAT
: Displays a list of items to track.

OFF, or 0
: Deactivates tracking of all variables. This value is the default.

ON, or 1
: Activates tracking of all variables. Tracking also activates whenever any specification changes.

DEL
: Removes the specified variable from the set of result items to track. If *Name* = ALL (default), all specifications are removed.

Name

The 32-character user-specified name.

Item, *Comp*

Predetermined output item and component label for valid elements. See the *ANSYS Elements Reference* for more information.

NODE

Valid node number (if *Key* = NSOL or ESOL), or valid real constant set number (if *Key* = PAIR) identifying a contact pair. Required input for NSOL, ESOL, and PAIR items.

ELEM

Valid element number for element results. Used for ESOL items. If *ELEM* is specified, then a node number that belongs to the element must also be specified in the *NODE* field.

SHELL

Valid labels are TOP, MID or BOT. This field can specify the location on shell elements for which to retrieve data. Used only for element nodal data (ESOL).

LAYER

Layer number (for layered elements only). Used only for element nodal data (ESOL).

Notes

The **NLHIST** command is a nonlinear diagnostics tool allowing you to monitor diagnostics results of interest in real time during solution.

You can track a maximum of 50 variables during solution. The specified result quantities are written to the file **Jobname.nlh**. Nodal results and contact results are written for every converged substep (irrespective of the **OUTRES** command setting) while element results are written only at time points specified via the **OUTRES** command. For time points where element results data is not available, a very small number is written instead. If the conditions for contact to be established are not satisfied, 0.0 will be written for contact results.

Results tracking is available only for a nonlinear structural analysis. All results are tracked in the Solution Coordinate System (that is, nodal results are in the nodal coordinate system and element results are in the element coordinate system).

The **Jobname.nlh** file is an ASCII file that lists each time point at which a converged solution occurs along with the values of the relevant result quantities.

The GUI option **Solution> Results tracking** provides an interface to define the result items to be tracked. The GUI also allows you to graph one or more variables against time or against other variables during solution. You can use the interface to graph or list variables from any **.nlh** file generated by the ANSYS program.

You can also track results during batch runs. Either access the ANSYS Launcher and select **File Tracking** from the **Tools** menu, or type **nlhist81** at the command line. Use the supplied file browser to navigate to your **Jobname.nlh** file, and click on it to invoke the tracking utilty. You can use this utilty to read the file at any time, even after the solution is complete (the data in the file must be formatted correctly).

Specifications that you set via the **NLHIST** command are not saved in the database (**.db** file).

Valid labels for **NSOL** solution quantities are:

- UX, UY, and UZ (displacements)
- ROTX, ROTY, and ROTZ (rotations)
- FX, FY, and FZ (reaction force)
- MX, MY, and MZ (reaction moment)

ETABLE items are not supported for **ESOL** items.

NLHIST - Valid ESOL Item and Component Labels

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.

Item	Comp	Description
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.

PAIR solution quantities are output on a "per contact pair" basis. As a consequence, the corresponding values listed in the **Jobname.nlh** file represent a minimum or a maximum over the associated contact pair, as detailed in the table below.

NLHIST - Valid PAIR Item and Component Labels

Item	Comp	Description
CONT	ELCN	If >0, number of contact elements in contact. Other values are interpreted as follows: 0 indicates the contact pair is in near-field contact status. -1 indicates the contact pair is in far-field contact status. -2 indicates that the contact pair is inactive (symmetric to asymmetric contact).
"	ELST	Number of contact elements in sticking contact status.
"	CNOS	Maximum chattering level.
"	PENE	Maximum penetration (or minimum gap). [1]
"	CLGP	Maximum closed gap.
"	SLID	Maximum total sliding distance.
"	ESLI	Maximum elastic slip distance.
"	KNMX	Maximum normal contact stiffness.
"	KTMX	Maximum tangential contact stiffness.
"	KNMN	Minimum normal contact stiffness.
"	KTMN	Minimum tangential contact stiffness.

Item	Comp	Description
"	PINB	Maximum pinball radius.
"	PRES	Maximum contact pressure.
"	SFRI	Maximum frictional stress.

1. For PENE, a positive value indicates a penetration, and a negative value indicates a gap. If the contact pair has a far-field contact status, penetration and gap are not available, and the value stored for PENE is the current pinball radius.

Menu Paths

Main Menu>Solution>Results Tracking

NLIST, *NODE1*, *NODE2*, *NINC*, *Lcoord*, *SORT1*, *SORT2*, *SORT3*

Lists nodes.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

NODE1, *NODE2*, *NINC*
List nodes from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL (default), *NODE2* and *NINC* are ignored and all selected nodes [**NSEL**] are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Lcoord
Coordinate listing key:

(blank)
List all nodal information

COORD
Suppress all but the XYZ coordinates (shown to a higher degree of accuracy than when displayed with all information).

SORT1
First item on which to sort. Valid item names are NODE, X, Y, Z, THXY, THYZ, THXZ

SORT2, *SORT3*
Second and third items on which to sort. Valid item names are the same as for *SORT1*.

Notes

Lists nodes in the active display coordinate system [**DSYS**]. Nodal coordinate rotation angles are also listed (relative to the global Cartesian coordinate system).

Node listing can be in a sorted order (ascending). *SORT2*, for example, will be carried out on nodes having equal values of *SORT1*.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>List Results>Sorted Listing>Sort Nodes
Utility Menu>List>Nodes

NLOG, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*, *FACTB*

Forms the natural log of a variable.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA
Reference number of the variable to be operated on.

--, --
Unused fields.

Name
Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --
Unused fields.

FACTA
Scaling factor applied to variable *IA* (defaults to 1.0).

FACTB
Scaling factor (positive or negative) applied to the operation (defaults to 1.0).

Notes

Forms the natural log of a variable according to the operation:

$$IR = FACTB * \mathrm{LN}(FACTA \times IA)$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Natural Log

NLOPT

Specifies "Nonlinear analysis options" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Nonlinear Options

NMODIF, *NODE*, *X*, *Y*, *Z*, *THXY*, *THYZ*, *THZX*

Modifies an existing node.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

NODE
: Modify coordinates of this node. If ALL, modify coordinates of all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

X, *Y*, *Z*
: Replace the previous coordinate values assigned to this node with these corresponding coordinate values. Values are interpreted in the active coordinate system (R, θ, Z for cylindrical; R, θ, Φ for spherical or toroidal). Leaving any of these fields blank retains the previous value(s).

THXY
: First rotation of nodal coordinate system about nodal Z (positive X toward Y). Leaving this field blank retains the previous value.

THYZ
: Second rotation of nodal coordinate system about nodal X (positive Y toward Z). Leaving this field blank retains the previous value.

THZX
: Third rotation of nodal coordinate system about nodal Y (positive Z toward X). Leaving this field blank retains the previous value.

Notes

Modifies an existing node. Nodal coordinate system rotation angles are in degrees and redefine any existing rotation angles. Nodes can also be redefined with the **N** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>RotateNode>By Angles
Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>Set of Nodes
Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>Single Node
Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>By Angles

---

NOCOLOR, *KEY*

Removes color from graphics displays.

DISPLAY: Set Up

MP ME ST DY <> PR EM <> FL PP ED

KEY
Color key:

0
Color the displays.

1
Do not color the displays.

2
Do not shade the displays.

Command Default

Color graphics displays (device dependent).

Menu Paths

It is part of the DISPLAY program.

NODES

Specifies "Nodes" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Nodes

/NOERASE

Prevents the screen erase between displays.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Command Default

Issue the command with no arguments to prevent automatic screen erase before creating the next display.

Notes

Preventing the normal screen erase between requested displays allows you to overlay multiple views.

Clearing the screen with the **ERASE** command (**Utility Menu> PlotCtrls> Erase Options> Erase screen**) active simply clears the display area. Subsequent replots will provide the cumulative plots previously generated by the **/NOERASE** command.

For 3-D devices, you can issue **/DV3D**,DELS to suppress repeated screen overlays and generate clear contour plots.

Use the **/ERASE** command to reactivate automatic screen erase.

For 3-D devices (**/SHOW**,3D), the model in all active windows will be the same, even if you issue a different display command (**NPLOT**, **EPLOT**, etc.) for each active window. Use the Multi-Plot command (**GPLOT**) to display different entities, in different windows, on 3-D devices.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Erase Options>Erase between Plots

/NOLIST

Suppresses the data input listing.

SESSION: List Controls

MP ME ST DY <> PR EM <> FL PP ED

Command Default

Issue the command with no arguments to suppress the data input listing.

Notes

Printout is suppressed until a **/GOLIST** command is read or the end of the listing is encountered.

This command is valid in any processor, but only within a batch run [**/BATCH**].

Menu Paths

This command cannot be accessed from a menu.

NOOFFSET, *Label*

Prevents the CDREAD command from offsetting specified data items

PREP7: Database

MP ME ST DY <> PR EM <> FL PP ED

Label

Specifies items not to be offset.

NODE
: Node numbers

ELEM
: Node numbers

KP
: Element numbers

LINE
: Keypoint numbers

AREA
: Area numbers

VOLU
: Volume numbers

MAT
: Material numbers

TYPE
: Element type numbers

REAL
: Real constant numbers

CSYS
: Coordinate system numbers

SECN
: Section numbers

CP
: Coupled set numbers

CE
: Constraint equation numbers

CLEAR
: All items will be offset

STATUS
: Shows which items are specified *not* to be offset.

Notes

The **NOOFFSET** command specifies data items not to be offset by a set of data read from a **CDREAD** command.

Menu Paths

This command cannot be accessed from a menu.

NOORDER, *Lab*

Re-establishes the original element ordering.

PREP7: Element Reordering

MP ME ST <> <> PR EM <> <> PP ED

Lab

Turns element reordering on or off.

ON (or blank)

Re-establishes original element ordering (default).

OFF

Original ordering is not used and program establishes its own ordering at the beginning of the solution phase.

Notes

If *Lab* = ON, the original element ordering is re-established and no automatic reordering occurs at the beginning of the solution phase. Use *Lab* = OFF only to remove the effect of a previous **NOORDER** command. This command affects only those elements that were defined up to the point that this command is issued. See the **WSORT** and **WAVES** commands for reordering.

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Element Reorder>Reset Elem Order

---

/NOPR

Suppresses the expanded interpreted input data listing.

SESSION: List Controls

MP ME ST DY <> PR EM <> FL PP ED

Command Default

Issuing this command with no arguments suppresses the interpreted data input print out.

Notes

Suppresses printout of interpreted input data, including information labeled as "Notes." When this printout is not suppressed, the data input to the analysis is echoed to the output file in an expanded format. Printout is suppressed until a **/GOPR** or **/GO** command is read.

Use of **/NOPR** is not recommended when the graphical user interface (GUI) is active. The GUI sometimes issues "hidden" **/NOPR** and **/GOPR** command sequences, which will countermand user-issued **/NOPR** commands, thus making the use of **/NOPR** in the GUI environment unpredictable.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Rigid Region
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Section
Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Smooth Data
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Smooth Data
Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts
Main Menu>Solution>Define Loads>Delete>Structural>Section
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Smooth Data
Main Menu>Solution>Loading Options>Smooth Data
Main Menu>Solution>Time Controls>Time Step Prediction
Main Menu>TimeHist Postpro>Smooth Data

NORA, *AREA*, *NDIR*

Rotates nodal coordinate systems to surface normal

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

AREA
: The area number containing the nodes to be rotated to their normals. If ALL, applies to all selected areas (see the **ASEL** command). If AREA = P, graphical picking is enabled.

NDIR
: Direction of the normal. If NDIR = -1, the nodal coordinate system is rotated in the opposite direction of the surface normal. The default is the same direction as the surface normal.

Notes

The **NORA** command rotates the X-axis of the nodal coordinate system to the surface normal. The rotated nodal coordinate systems may be displayed through the **/PSYMB** command. In case multiple areas are selected, there could be conflicts at the boundaries. If a node belongs to two areas that have a different normal, its nodal coordinate system will be rotated to the area normal with the lowest number. You can use the **AREVERSE** and **ANORM** commands to rotate the surface normals in the appropriate direction. Keep the following in mind when using the **NORA** command:

- If the nodal coordinate system is parallel to the global Cartesian system, it is not displayed through the **/PSYMB** command.

- Previously specified rotation on the selected nodes are overridden.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Areas

---

NORL, *LINE*, *AREA*, *NDIR*

Rotates nodal coordinate systems perpendicular to line normal

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

LINE
: Line number containing the nodes to be rotated. If ALL, applies to all selected lines (see the **LSEL** command). If LINE = P, graphical picking is enabled.

AREA
: The area number containing the selected lines. The normal of the line(s) selected is supposed to lie on this area. Defaults to the lowest numbered selected area containing the line number.

NDIR
: Direction of the normal. If NDIR = -1, the nodal coordinate system is rotated in the opposite direction of the line normal. The default is the same direction as the surface normal.

Notes

The **NORL** command rotates the X-axis of the nodal coordinate perpendicular to the line normal. The rotated nodal coordinate systems may be displayed through the **/PSYMB** command. In case multiple lines are selected, there could be conflicts at the boundaries. If a node belongs to two lines that have a different normal, its nodal coordinate system will be rotated to the line normal with the lowest number. Keep the following in mind when using the **NORL** command:

- If the nodal coordinate system is parallel to the global Cartesian system, it is not displayed through the **/PSYMB** command.
- Previously specified rotation on the selected nodes are overridden.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Lines
Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>with Area

---

/NORMAL, *WN*, *KEY*

Allows displaying area elements by top or bottom faces.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

WN
: Window number (or ALL) to which command applies (defaults to 1).

KEY
Display key:

0
No face distinction.

1
Show only area elements having their positive normals directed toward the viewing point.

1
Show only area elements having their positive normals directed away from the viewing point.

Command Default

Do not distinguish between top face and bottom face elements.

Notes

/NORMAL allows you to select area elements and area plots by the top or bottom faces. It is useful for checking the normal directions on shell elements. The positive normal (element Z direction) is defined by the right-hand rule following the node I, J, K, L input direction. This command is available only with raster or hidden-line displays, for WIN32 or X11 2-D displays only.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Shell Normals

NPLOT, *KNUM*
Displays nodes.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

KNUM
Node number key:

0
No node numbers on display.

1
Include node numbers on display. See also **/PNUM** command.

Notes

Produces a node display. Only selected nodes [**NSEL**] are displayed. Elements need not be defined. See the **DSYS** command for display coordinate system.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Areas
Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Lines
Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>with Area
Utility Menu>Plot>Nodes

NPRINT, *N*

Defines which time points stored are to be listed.

POST26: Listing

MP ME ST DY <> PR EM <> FL PP ED

N

List data associated with every *N* time (or frequency) point(s), beginning with the first point stored (defaults to 1).

Command Default

List all stored points.

Notes

Defines which time (or frequency) points within the range stored are to be listed.

Menu Paths

Main Menu>TimeHist Postpro>Settings>List

NREAD, *Fname*, *Ext*, --

Reads nodes from a file.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext

Filename extension (8 character maximum).

The extension defaults to NODE if *Fname* is blank.

--

Unused field

Notes

The read operation is not necessary in a standard ANSYS run but is provided as a convenience to users wanting to read a coded node file, such as from another mesh generator or from a CAD/CAM program. Data should be formatted as produced with the **NWRITE** command. The element types [**ET**] must be defined before the file is read so that the file may be read properly. Only nodes that are within the node range specified with the **NRRANG** command are read from the file. Duplicate nodes already in the database will be overwritten. The file is rewound before and after reading. Reading continues until the end of the file.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>Read Node File

NREFINE, *NN1*, *NN2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

Refines the mesh around specified nodes.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NN1, *NN2*, *NINC*

Nodes (*NN1* to *NN2* in increments of *NINC*) around which the mesh is to be refined. *NN2* defaults to *NN1*, and *NINC* defaults to 1. If *NN1* = ALL, *NN2* and *NINC* are ignored and all selected nodes are used for refinement. If *NN1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NN1* (*NN2* and *NINC* are ignored).

LEVEL

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

DEPTH

Depth of mesh refinement in terms of number of elements outward from the indicated nodes (defaults to 1).

POST

Type of postprocessing to be done after element splitting, in order to improve element quality:

OFF

No postprocessing will be done.

SMOOTH

Smoothing will be done. Node locations may change.

CLEAN

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

RETAIN

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The ANSYS program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

ON
The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

OFF
The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

Notes

NREFINE performs local mesh refinement around the specified nodes. By default, the indicated elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

NREFINE refines all area elements and tetrahedral volume elements that are adjacent to the specified nodes. Any volume elements that are adjacent to the specified nodes, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes [**IC**], coupled nodes [**CP** family of commands], constraint equations [**CE** family of commands], or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. For additional restrictions on mesh refinement, see Revising Your Model in the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Nodes

NRLSUM, *SIGNIF*, *Label*

Specifies the Naval Research Laboratory (NRL) sum mode combination method.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

SIGNIF
Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single point, multi-point, or DDAM response (**SPOPT**,SPRS, MPRS or DDAM), the significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT**,PSD.)

Label
Label identifying the combined mode solution output.

DISP
Displacement solution (default). Displacements, stresses, forces, etc., are available.

VELO
Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

ACEL
Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

Notes

This command is also valid in PREP7. This mode combination method is usually used for **SPOPT**,DDAM.

Product Restrictions:

NRLSUM is not allowed in ANSYS Professional.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine

---

NROPT, *Option*, --, *Adptky*

Specifies the Newton-Raphson options in a static or full transient analysis.

SOLUTION: Nonlinear Options

MP ME ST <> <> <> EM <> <> PP ED

Option
Option key:

AUTO
Let the program choose the option (default).

FULL
Use full Newton-Raphson.

MODI
Use modified Newton-Raphson.

INIT
Use the previously computed matrix (initial-stiffness).

UNSYM
Use full Newton-Raphson with unsymmetric matrices of elements where the unsymmetric option exists (see Newton-Raphson Option in the *ANSYS Structural Analysis Guide* for more information).

--
Unused field.

Adptky
Adaptive descent key:

ON
Use adaptive descent (default if frictional contact exists; if one of the elements CONTAC12 or CONTAC52 is present; or if plasticity exists and one of the elements PIPE20, BEAM23, BEAM24, or PIPE60 is present). Explicit ON is valid only if *Option* = FULL.

OFF
Do not use adaptive descent (default in all other cases).

Notes

The default values given for this command assume **SOLCONTROL**,ON (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL**,ON and **SOLCONTROL**,OFF.

Indicates the Newton-Raphson option used to solve the nonlinear equations in a static or full transient analysis. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

NROTAT, *NODE1*, *NODE2*, *NINC*

Rotates nodal coordinate systems into the active system.

PREP7: Nodes

MP ME ST <> <> PR EM <> FL PP ED

NODE1, *NODE2*, *NINC*
Rotate nodes from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and all selected nodes [**NSEL**] are rotated. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Command Default

No automatic nodal rotation.

Notes

Rotates nodal coordinate systems into the active coordinate system. Nodal coordinate systems may be automatically rotated into the active (global or local) coordinate system as follows: Rotations in Cartesian systems will have nodal x directions rotated parallel to the Cartesian X direction. Rotations in cylindrical, spherical or toroidal systems will have the nodal x directions rotated parallel to the R direction. Nodes at (or near) a zero radius location should not be rotated. Nodal coordinate directions may be displayed [**/PSYMB**]. Nodal forces and constraints will also appear rotated when displayed if the nodal coordinate system is rotated. For FLOTRAN analyses, nodal coordinate systems should only be rotated parallel to the global Cartesian system.

ANSYS LS-DYNA (explicit dynamics) does not support the **NROTAT** command. If you have rotated nodes in the implicit phase of an implicit-to-explicit sequential solution, you must rotate the nodes back to the global Cartesian direction before switching from implicit to explicit elements (**ETCHG**,ITE). Use the **EDNROT** command in the explicit run to maintain the same displacement constraints as were used on rotated nodes in the implicit run.

Note — When the nodal coordinate systems are defined, they remain parallel to the global Cartesian system unless subsequently rotated.

Previously specified rotations on the specified nodes are overridden.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>RotateNode>To Active CS
Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Active CS

NRRANG, *NMIN*, *NMAX*, *NINC*

Specifies the range of nodes to be read from the node file.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

NMIN, *NMAX*, *NINC*

Node range is defined from *NMIN* (defaults to 1) to *NMAX* (defaults to 99999999) in steps of *NINC* (defaults to 1).

Notes

Defines the range of nodes to be read [**NREAD**] from the node file. Also implies an element range since only elements fully attached to these nodes will be read from the element file.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>Read Node File

NSCALE, *INC*, *NODE1*, *NODE2*, *NINC*, *RX*, *RY*, *RZ*

Generates a scaled set of nodes from a pattern of nodes.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

INC

Do this scaling operation one time, incrementing all nodes in the given pattern by *INC*. If *INC* = 0, nodes will be redefined at the scaled locations.

NODE1, *NODE2*, *NINC*

Scale nodes from pattern of nodes beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and pattern is all selected nodes [**NSEL**]. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

RX, *RY*, *RZ*

Scale factor ratios. Scaling is relative to the origin of the active coordinate system (RR, Rθ, RZ for cylindrical, RR, Rθ, RΦ for spherical or toroidal). If absolute value of ratio > 1.0, pattern is enlarged. If < 1.0, pattern is reduced. Ratios default to 1.0 (each).

Notes

Generates a scaled pattern of nodes from a given node pattern. Scaling is done in the active coordinate system. Nodes in the pattern may have been generated in any coordinate system.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Nodes>Scale & Copy
Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>Scale & Move
Main Menu>Preprocessor>Modeling>Operate>Scale>Nodes>Scale & Copy
Main Menu>Preprocessor>Modeling>Operate>Scale>Nodes>Scale & Move

---

NSEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS*

Selects a subset of nodes.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

ALL
Restore the full set.

NONE
Unselect the full set.

INVE
Invert the current set (selected becomes unselected and vice versa).

STAT
Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Item
Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to NODE.

Comp
Component of the item (if required). Valid component labels are shown in the table below.

VMIN
Minimum value of item range. Ranges are node numbers, set numbers, coordinate values, load values, or result values as appropriate for the item. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored).

VMAX
Maximum value of item range. *VMAX* defaults to *VMIN* for input values. For result values, *VMAX* defaults to infinity if *VMIN* is positive, or to zero if *VMIN* is negative. If *VMIN* = *VMAX*, a tolerance of ±0.005x *VMIN* is used, or ±1.0E-6 if *VMIN*=0.0. If *VMAX* ≠ *VMIN*, a tolerance of 1.0E-8x (*VMAX*-*VMIN*) is used.

VINC
Value increment within range. Used only with integer ranges (such as for node and set numbers). Defaults to 1. *VINC* cannot be negative.

KABS
Absolute value key:

0
Check sign of value during selection.

1
Use absolute value during selection (sign ignored).

Command Default

All nodes are selected.

Notes

Selects a subset of nodes. For example, to select a new set of nodes based on node numbers 1 through 7, use **NSEL**,S,NODE,,1,7. The subset is used when the ALL label is entered (or implied) on other commands, such as **NLIST**,ALL. Only data identified by node number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

When selecting nodes by results, the full graphics value is used, regardless of whether PowerGraphics is on.

Solution result data consists of two types, 1) nodal degree of freedom--results initially calculated at the nodes (such as displacement, temperature, pressure, etc.), and 2) element--results initially calculated elsewhere (such as at an element integration point or thickness location) and then recalculated at the nodes (such as stresses, strains, etc.). Various element results also depend upon the recalculation method and the selected results location [**AVPRIN**, **RSYS**, **FORCE**, **LAYER** and **SHELL**].

You must have all the nodes (corner and midside nodes) on the external face of the element selected to use *Item* = EXT.

This command is valid in any processor.

NSEL - Valid Item and Component Labels

Valid Item and Component Labels NSEL, *Type,Item,Comp,VMIN,VMAX,VINC,KABS*

Valid item and component labels for input values are:

Item	Comp	Description
NODE		Node number.

Valid Item and Component Labels NSEL, *Type,Item,Comp,VMIN,VMAX,VINC,KABS*

Valid item and component labels for input values are:

Item	Comp	Description
EXT		Nodes on exterior of selected elements (ignore remaining fields).
LOC	X, Y, Z	X, Y, or Z location in the active coordinate system.
ANG	XY, YZ, ZX	THXY, THYZ, or THZX rotation angle.
M		Master node number.
CP		Coupled set number.
CE		Constraint equation set number.
D	U	Any of X, Y, or Z structural displacements. Amplitude only, if complex.
"	UX, UY, UZ	X, Y, or Z structural displacement. Amplitude only, if complex.
"	ROT	Any of X, Y, or Z structural rotations. Amplitude only, if complex.
"	ROTX, ROTY, ROTZ	X, Y, or Z structural rotation. Amplitude only, if complex.
"	TEMP, TBOT, TE2, TE3, . . ., TTOP	Temperature.
"	PRES	Pressure.
"	VOLT	Electric potential.
"	MAG	Magnetic scalar potential.
"	V	Any of X, Y, or Z fluid velocities.
"	VX, VY, VZ	X, Y, or Z fluid velocity.
"	A	Any of X, Y, or Z magnetic vector potentials. Amplitude only, if complex.
"	AX, AY, AZ	X, Y, or Z magnetic vector potential. Amplitude only, if complex.
"	CURR	Current.
"	EMF	Electromotive force drop.
"	ENKE, ENDS	Turbulent kinetic energy or energy dissipation (FLOTRAN).
F	F	Any of X, Y, or Z structural forces. Amplitude only, if complex.
"	FX, FY, FZ	X, Y, or Z structural force. Amplitude only, if complex.
"	M	Any of X, Y, or Z structural moments. Amplitude only, if complex
"	MX, MY, MZ	X, Y, or Z structural moment. Amplitude only, if complex.
"	HEAT, HBOT, HE2, HE3, . . ., HTOP	Heat flow.
"	FLOW	Fluid flow.
"	AMPS	Current flow.
"	FLUX	Magnetic flux.
"	CSG	Any of X, Y, or Z magnetic current segment components. Amplitude only, if complex.
"	CSGX, CSGY,	X, Y, or Z magnetic current segment component. Amplitude only, if complex.
	CSGZY,	if complex.
"	CHRG	Electric charge.
"	CHRGD	Electric charge density.
BF	TEMP	Nodal temperature.

Valid Item and Component Labels NSEL, *Type,Item,Comp,VMIN,VMAX,VINC,KABS*

Valid item and component labels for input values are:

Item	Comp	Description
"	FLUE	Nodal fluence.
"	HGEN	Nodal heat generation rate.
"	JS	Any of X, Y, or Z current densities. Amplitude only, if complex.
"	JSX, JSY, JSZ	X, Y, or Z current density. Amplitude only, if complex.
"	MVDI	Magnetic virtual displacements flag.

NSEL - Valid Item and Component Labels for Nodal DOF Result Values

Item	Comp	Description
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum.
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum.
TEMP		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum.
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy (FLOTRAN).
ENDS		Turbulent energy dissipation (FLOTRAN).

NSEL - Valid Item and Component Labels for Element Result Values

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1,2,3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1,2,3	Principal plastic strain.
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1,2,3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.

Item	Comp	Description
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
CONT	STAT	Contact status. 3-closed and sticking, 2-closed and sliding, 1-open but near contact, 0-open and not near contact.
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component magnetic forces or vector sum.
TOPO		Densities used for topological optimization. This applies to nodes attached to the following types of elements: PLANE2, PLANE82, SOLID92, SHELL93, SOLID95.

NSEL - Valid Item and Component Labels for FLOTRAN Nodal Result Values

Item	Comp	Description
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
CMUV		Turbulent viscosity coefficient.

Item	Comp	Description
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Section
Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements
Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts
Main Menu>Solution>Define Loads>Delete>Structural>Section
Utility Menu>Select>Entities

NSLA, *Type*, *NKEY*

Selects those nodes associated with the selected areas.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of node select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

NKEY
Specifies whether only interior area nodes are to be selected:

0
Select only nodes interior to selected areas.

1
Select all nodes (interior to area, interior to lines, and at keypoints) associated with the selected areas.

Notes

Valid only if the nodes were generated by an area meshing operation [**AMESH**, **VMESH**] on a solid model that contains the selected areas.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

NSLE, *Type*, *NodeType*, *Num*

Selects those nodes attached to the selected elements.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type

Label identifying the type of node select:

S
: Select a new set (default).

R
: Reselect a set from the current set.

A
: Additionally select a set and extend the current set.

U
: Unselect a set from the current set.

NodeType

Label identifying type of nodes to consider when selecting:

ALL
: Select all nodes of the selected elements (default).

ACTIVE
: Select only the active nodes. An active node is a node that contributes DOFs to the model.

INACTIVE
: Select only inactive nodes (such as orientation or radiation).

CORNER
: Select only corner nodes.

MID
: Select only midside nodes.

POS
: Select nodes in position *Num*.

FACE
: Select nodes on face *Num*.

Num

Position or face number for *Nodes* = POS or FACE.

Notes

NSLE selects *NodeType* nodes attached to the currently-selected set of elements. Only nodes on elements in the currently-selected element set can be selected.

Note — When using degenerate hexahedral elements, **NSLE**, *U*,*CORNER* and **NSLE**,*S*,*MID* will not select the same set of nodes because some nodes appear as both corner and midside nodes.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking
Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts
Utility Menu>Select>Entities

NSLK, *Type*

Selects those nodes associated with the selected keypoints.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of node select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

Notes

Valid only if the nodes were generated by a keypoint meshing operation [**KMESH**, **LMESH**, **AMESH**, **VMESH**] on a solid model that contains the selected keypoints.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

NSLL, *Type*, *NKEY*

Selects those nodes associated with the selected lines.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of node select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

NKEY
Specifies whether only interior line nodes are to be selected:

0
Select only nodes interior to selected lines.

1
Select all nodes (interior to line and at keypoints) associated with the selected lines.

Notes

Valid only if the nodes were generated by a line meshing operation [**LMESH**, **AMESH**, **VMESH**] on a solid model that contains the associated lines.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

NSLV, *Type*, *NKEY*
Selects those nodes associated with the selected volumes.

DATABASE: Selecting
MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of node select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

NKEY
Specifies whether only interior volume nodes are to be selected:

0
Select only nodes interior to selected volumes.

1
Select all nodes (interior to volume, interior to areas, interior to lines, and at keypoints) associated with the selected volumes.

Notes

Valid only if the nodes were generated by a volume meshing operation [**VMESH**] on a solid model that contains the selected volumes.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

NSMOOTH, *NPASS*

Smooths selected nodes among selected elements.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

NPASS
Number of smoothing passes. Defaults to 3.

Notes

Repositions each selected node at the average position of its immediate neighbors on the selected elements. The node positions converge after some number of smoothing passes. For some initial conditions, *NPASS* may need to be much larger than 3. If the boundary of a mesh is to be undisturbed (usually desirable), the boundary nodes should be unselected before issuing **NSMOOTH**.

Menu Paths

This command cannot be accessed from a menu.

NSOL, *NVAR*, *NODE*, *Item*, *Comp*, *Name*

Specifies nodal data to be stored from the results file.

POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

NVAR
Arbitrary reference number or name assigned to this variable. Variable numbers can be 2 to *NV* (**NUMVAR**) while the name can be an eight byte character string. Overwrites any existing results for this variable.

NODE
Node for which data are to be stored.

Item
Label identifying the item. Valid item labels are shown in the table below. Some items also require a component label.

Comp
Component of the item (if required). Valid component labels are shown in the table below.

Name
Thirty-two character name identifying the item on printouts and displays. Defaults to a label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Notes

Stores nodal degree of freedom and solution results in a variable. For more information, see Data Interpreted in the Nodal Coordinate System in the *ANSYS Modeling and Meshing Guide*.

NSOL - Valid Item and Component Labels

Valid Item and Component Labels NSOL, *NVAR,NODE,Item,Comp,Name*

Valid item and component labels for nodal degree of freedom results are:

Item	Comp	Description
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.
TEMP[1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity in a fluid analysis, or X, Y, or Z nodal velocity in an ANSYS structural transient dynamic analysis (**ANTYPE**,TRANS).
A	X, Y, Z	X, Y, or Z magnetic vector potential in an electromagnetic analysis, or X, Y, or Z acceleration in an ANSYS structural transient dynamic analysis (**ANTYPE**,TRANS).
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy.
ENDS		Turbulent energy dissipation.

NSOL - Valid Item and Component Labels for FLOTRAN Nodal Results

Item	Comp	Description
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.

Item	Comp	Description
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
CMUV		Turbulent viscosity coefficient.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.
RDFL		Radiation heat flux.

NSOL - Valid Item and Component Labels for ANSYS LS-DYNA Nodal Results

Item	Comp	Description
U	X, Y, Z	X, Y, or Z nodal displacements.
ROT	X, Y, Z	X, Y, or Z nodal rotations.
V	X, Y, Z	X, Y, or Z nodal velocities.
A	X, Y, Z	X, Y, or Z nodal accelerations.

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . ., TTOP instead of TEMP.

Menu Paths

Main Menu>Drop Test>Time History>Graph Variables
Main Menu>Drop Test>Time History>List Variables
Main Menu>TimeHist Postpro>Define Variables
Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

NSORT, *Item*, *Comp*, *ORDER*, *KABS*, *NUMB*, *SEL*

Sorts nodal data.

POST1: Results

MP ME ST DY <> PR EM <> FL PP ED

Item
Label identifying the item to be sorted on. Valid item labels are shown in the table below. Some items also require a component label.

Comp
Component of the item (if required). Valid component labels are shown in the table below.

ORDER
Order of sort operation:

0
Sort into descending order.

1
Sort into ascending order.

KABS
Absolute value key:

0
Sort according to real value.

1
Sort according to absolute value.

NUMB
Number of nodal data records to be sorted in ascending or descending order (*ORDER*) before sort is stopped (remainder will be in unsorted sequence) (defaults to all nodes).

SEL
Allows selection of nodes in the sorted field.

(blank)
No selection (default).

SELECT
Select the nodes in the sorted list.

Command Default

Use ascending node number order.

Notes

Values are in the active coordinate system [**CSYS** for input data or **RSYS** for results data]. Various element results also depend upon the recalculation method and the selected results location [**AVPRIN**, **RSYS**, **SHELL**, **ESEL**, and **NSEL**]. If simultaneous load cases are stored, the last sorted sequence formed from any load case applies to all load cases. Use **NUSORT** to restore the original order. This command is not valid with PowerGraphics.

NSORT - Valid Item and Component Labels

Valid Item and Component Labels NSORT, *Item,Comp,ORDER,KABS,NUMB,SEL*

Valid item and component labels for input values are:

Item	Comp	Description
LOC	X, Y, Z	X, Y, or Z location.
ANG	XY, YZ, ZX	THXY, THYZ, or THZX rotation angle.

NSORT - Valid Item and Component Labels for Nodal DOF Result Values

Item	Comp	Description
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum.
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum.
TEMP		Temperature (includes TEMP, TBOT, TE2, TE3, . . ., TTOP values).
PRES		Pressure.

Item	Comp	Description
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum.
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy (FLOTRAN).
ENDS		Turbulent energy dissipation (FLOTRAN).

NSORT - Valid Item and Component Labels for Element Result Values

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2,3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPLS	X, Y, Z, XY, YZ, XZ	Component large strain.
"	1, 2, 3	Principal large strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
CONT	STAT	Contact status. 3-closed and sticking, 2-closed and sliding, 1-open but near contact, 0-open and not near contact.
"	PENE	Contact penetration.

Item	Comp	Description
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component magnetic forces or vector sum.
TOPO		Densities used for topological optimization. This applies to nodes attached to the following types of elements: PLANE2, PLANE82, SOLID92, SHELL93, SOLID95.

NSORT - Valid Item and Component Labels for FLOTRAN Nodal Result Values

Item	Comp	Description
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.

Menu Paths

Main Menu>General Postproc>List Results>Sorted Listing>Sort Nodes
Utility Menu>Parameters>Get Scalar Data

NSTORE, *TINC*

Defines which time points are to be stored.

POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

TINC

Store data associated with every *TINC* time (or frequency) point(s), within the previously defined range of *TMIN* to *TMAX* [**TIMERANGE**]. (Defaults to 1)

Command Default

Store every point.

Notes

Defines which time (or frequency) points within the range are to be stored.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Data

NSUBST, *NSBSTP*, *NSBMX*, *NSBMN*, *Carry*

Specifies the number of substeps to be taken this load step.

SOLUTION: Load Step Options

MP ME ST <> <> PR EM <> <> PP ED

NSBSTP

Number of substeps to be used for this load step (i.e., the time step size or frequency increment). If automatic time stepping is used [**AUTOTS**], *NSBSTP* defines the size of the first substep. If **SOLCONTROL**,ON and contact elements TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, or CONTA174 are used, defaults to 1 or 20 substeps, depending on the physics of the problem. If **SOLCONTROL**,ON and none of these contact elements are used, defaults to 1 substep. If **SOLCONTROL**,OFF, defaults to the previously specified value (or 1, if there is no previously specified value).

NSBMX

Maximum number of substeps to be taken (i.e., the minimum time step size) if automatic time stepping is used. If **SOLCONTROL**,ON, ANSYS determines the default depending on the physics of the problem. If **SOLCONTROL**,OFF, defaults to the previously specified value (or *NSBSTP*, if there is no previously specified value).

NSBMN

Minimum number of substeps to be taken (i.e., the maximum time step size) if automatic time stepping is used. If **SOLCONTROL**,ON, ANSYS determines the default depending on the physics of the problem. If **SOLCONTROL**,OFF, defaults to the previously specified value (or 1, if there is no previously specified value).

Carry

Time step carryover key (ANSYS determines the default depending on the physics of the problem):

OFF

Use *NSBSTP* to define time step at start of each load step.

ON
Use final time step from previous load step as the starting time step (if automatic time stepping is used).

If **SOLCONTROL**,ON, ANSYS determines the default depending on the physics of the problem. If **SOLCONTROL**,OFF, defaults to OFF.

Notes

See **DELTIM** for an alternative input. It is recommended that all fields of this command be specified for solution efficiency and robustness.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Freq and Substps
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substps
Main Menu>Solution>Analysis Type>Sol'n Controls>Basic
Main Menu>Solution>Load Step Opts>Time/Frequenc>Freq and Substps
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substps

---

NSVR, *ITYPE*, *NSTV*

Defines the number of variables for user-programmable element options.

PREP7: Element Type

MP ME ST DY <> <> <> <> <> PP <>

ITYPE
Element type number as defined on the **ET** command.

NSTV
Number of extra state variables to save (must be no more than 840).

Command Default

No extra variables are saved.

Notes

Defines the number of extra variables that need to be saved for user-programmable (system-dependent) element options, e.g., material laws through user subroutine USERPL. *ITYPE* must first be defined with the **ET** command.

Menu Paths

This command cannot be accessed from a menu.

NSYM, *Ncomp*, *INC*, *NODE1*, *NODE2*, *NINC*

Generates a reflected set of nodes.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

Ncomp

Symmetry key:

X
: X (or R) symmetry (default).

Y
: Y (or θ) symmetry.

Z
: Z (or Φ) symmetry.

INC

Increment all nodes in the given pattern by *INC* to form the reflected node pattern.

NODE1, *NODE2*, *NINC*

Reflect nodes from pattern beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and pattern is all selected nodes [**NSEL**]. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Notes

Generates nodes from a given node pattern by a symmetry reflection. Reflection is done in the active coordinate system by changing a particular coordinate sign. Nodes in the pattern may have been generated in any coordinate system. Nodal rotation angles are not reflected.

Symmetry reflection may be used with any node pattern, in any coordinate system, as many times as desired. Reflection is accomplished by a coordinate sign change (in the active coordinate system). For example, an X-reflection in a Cartesian coordinate system generates additional nodes from a given pattern, with a node increment added to each node number, and an X coordinate sign change. An R-reflection in a cylindrical coordinate system gives a reflected "radial" location by changing the "equivalent" Cartesian (i.e., the Cartesian system with the same origin as the active cylindrical system) X and Y coordinate signs. An R-reflection in a spherical coordinate system gives a reflected "radial" location by changing the equivalent Cartesian X, Y, and Z coordinate location signs. Nodal coordinate system rotation angles are not reflected.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Nodes

/NUMBER, *NKEY*

Specifies whether numbers, colors, or both are used for displays.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

NKEY
Numbering style:

0
Color (terminal dependent) the numbered items and show numbers.

1
Color the numbered items. Do not show the numbers.

2
Show the numbers. Do not color the items.

1
Do not color the items or show the numbers. For contour plots, the resulting display will vary (see below).

Notes

Specifies whether numbers, colors, or both are used for numbering displays [**/PNUM**] of nodes, elements, keypoints, lines, areas, and volumes.

Shading is also available for terminals configured with more than 4 color planes [**/SHOW**]. Color automatically appears for certain items and may be manually controlled (off or on) for other items.

When you suppress color (*NKEY* = -1) your contour plots will produce different results, depending on your graphics equipment. For non-3-D devices (X11, Win32, etc.) your contour plot will be white (no color). For 3-D devices, such as OpenGL, the resulting display will be in color.

The following items are automatically given discrete colors: Boundary condition symbols [**/PBC**], curves on graph displays, and distorted geometry on postprocessing displays. Contour lines in postprocessing displays are automatically colored based upon a continuous, rather than a discrete, spectrum so that red is associated with the highest contour value. On terminals with raster capability [**/SHOW**], the area between contour lines is filled with the color of the higher contour.

Explicit entity colors or the discrete color mapping may be changed with the **/COLOR** command.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Numbering

NUMCMP, *Label*

Compresses the numbering of defined items.

PREP7: Database

MP ME ST DY <> PR EM <> FL PP ED

Label

Items to be compressed:

NODE
Node numbers

ELEM
Element numbers

KP
Keypoint numbers

LINE
Line numbers

AREA
Area numbers

VOLU
Volume numbers

MAT
Material numbers

TYPE
Element type numbers

REAL
Real constant numbers

CP
Coupled set numbers

CE
Constraint equation numbers

ALL
All item numbers

Notes

The **NUMCMP** command effectively compresses out unused item numbers by renumbering all the items, beginning with one and continuing throughout the model. The renumbering order follows the initial item numbering order (that is, compression lowers the maximum number by "sliding" numbers down to take advantage of unused or skipped numbers). All defined items are renumbered, regardless of whether or not they are actually used or selected. Applicable related items are also checked for renumbering as described for the merge operation [**NUMMRG**].

Compressing material numbers [**NUMCMP**,ALL or **NUMCMP**,MAT] does *not* update the material number referenced by any of the following:

- A temperature-dependent convection or surface-to-surface radiation load [**SF**, **SFE**, **SFL**, **SFA**]

- Real constants for multi-material elements (PIPE17, SOLID46, SOLID65, SHELL91, SHELL99)
- Section information for multi-material elements (SHELL131, SHELL132, SHELL181, BEAM188, BEAM189, SHELL208, and SHELL209). See also the **SECDATA** and **SECWRITE** command descriptions.

Compression is usually not required unless memory space is limited and there are large gaps in the numbering sequence.

You cannot access this command for models that have been imported from IGES files using the FACETED translator (see the **IOPTN** command).

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Compress Numbers

NUMEXP, *NUM*, *BEGRNG*, *ENDRNG*, *Elcalc*

Specifies solutions to be expanded from reduced analyses.

SOLUTION: Load Step Options

MP ME ST <> <> PR <> <> <> PP ED

NUM

The number of solutions to expand for the current load step. This value is required.

Num

Number of solutions to expand.

ALL

Expand all substeps between *BEGRNG* and *ENDRNG* (provided that *ENDRNG* > 0). If *BEGRNG* and *ENDRNG* have no specified values, this option expands *all* substeps for the current load step.

BEGRNG, *ENDRNG*

Beginning and ending time (or frequency) range for expanded solutions. The default is 0 for both values.

Elcalc

The element-calculation key:

YES

Calculate element results, nodal loads, and reaction loads. This value is the default.

NO

Do not calculate these items.

Command Defaults

Issuing this command with no arguments is invalid. You must specify the number of solutions, or all solutions, to expand (*NUM*). The default value for both the beginning (*BEGRNG*) and ending (*ENDRNG*) time or frequency is 0. The default behavior of the command is to calculate element results, nodal loads, and reaction loads (*Elcalc* = YES).

Notes

Specifies a range of solutions to be expanded from analyses that use reduced or mode superposition methods (**ANTYPE**,HARMIC, TRANS, or SUBSTR).

For **ANTYPE**,TRANS, *NUM*, evenly spaced solutions are expanded between time *BEGRNG* and time *ENDRNG*.

For **ANTYPE**,HARMIC, *NUM*, evenly spaced solutions are expanded between frequency *BEGRNG* and frequency *ENDRNG*.

The first expansion in all cases is done at the first point beyond *BEGRNG* (that is, at *BEGRNG* + (*ENDRNG* - *BEGRNG*) / *NUM*)).

For a single expansion of a solution, or for multiple expansions when the solutions are not evenly spaced (such as in a mode superposition harmonic analysis with the cluster option), ANSYS recommends issuing one or more **EXPSOL** commands.

The **NUMEXP** command is invalid in these cases:

- In a substructing analysis (**ANTYPE**,*SUBST*) when a triangularized matrix file (the **.TRI** file generated by the frontal solver or the **.LN22** file generated by the sparse solver) does not exist, causing ANSYS to employ the full-resolve method.
- If the full-resolve option is selected using the **SEOPT** command.

In both situations, use the **EXPSOL** command to perform a single expansion for each solution desired.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's
Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's

---

NUMMRG, *Label*, *TOLER*, *GTOLER*, *Action*, *Switch*

Merges coincident or equivalently defined items.

PREP7: Database

MP ME ST DY <> PR EM <> FL PP ED

Label

Items to be merged:

NODE
: Nodes

ELEM
: Elements

KP
: Keypoints (will also merge lines, areas, and volumes)

MAT
: Materials

TYPE
Element types

REAL
Real constants

CP
Coupled sets

CE
Constraint equations

ALL
All items

TOLER
Range of coincidence. For *Label* = NODE and KP, defaults to 1.0E-4 (based on maximum Cartesian coordinate difference between nodes or keypoints). For *Label* = MAT, REAL, and CE, defaults to 1.0E-7 (based on difference of the values normalized by the values). Only items within range are merged. (For keypoints attached to lines, further restrictions apply. See the *GTOLER* field and *Merging Solid Model Entities* below.)

GTOLER
Global solid model tolerance -- used only when merging keypoints attached to lines. If specified, *GTOLER* will override the internal relative solid model tolerance. See *Merging Solid Model Entities* below.

Action
Specifies whether to merge or select coincident items.

SELE
Select coincident items but do not merge. *Action* = SELE is only valid for *Label* = NODE.

(Blank)
Merge the coincident items (default).

Switch
Specifies whether the lowest or highest numbered coincident item is retained after the merging operation. This option does not apply to keypoints; i.e., for *Label* = KP, the lowest numbered keypoint is retained regardless of the *Switch* setting.

LOW
Retain the lowest numbered coincident item after the merging operation (default).

HIGH
Retain the highest numbered coincident item after the merging operation.

Notes

After issuing the command, the area and volume sizes (**ASUM** and **VSUM**) may give slightly different results. In order to obtain the same results as before, use **/FACET**, **/NORMAL**, and **ASUM** / **VSUM**.

The merge operation is useful for tying separate, but coincident, parts of a model together. If not all items are to be checked for merging, use the select commands (**NSEL**, **ESEL**, etc.) to select items. Only selected items are included in the merge operation for nodes, keypoints, and elements.

By default, the merge operation retains the lowest numbered coincident item. Higher numbered coincident items are deleted. Set *Switch* to HIGH to retain the highest numbered coincident item after the merging operation. Applicable related items are also checked for deleted item numbers and if found, are replaced with the

retained item number. For example, if nodes are merged, element connectivities (except superelements), mesh item range associativity, coupled degrees of freedom, constraint equations, master degrees of freedom, gap conditions, degree of freedom constraints, nodal force loads, nodal surface loads, and nodal body force loads are checked. Merging material numbers [**NUMMRG**,ALL or **NUMMRG**,MAT] does *not* update the material number referenced:

1. by temperature-dependent film coefficients as part of convection load or a temperature-dependent emissivity as part of a surface-to-surface radiation load [**SF**, **SFE**, **SFL**, **SFA**]
2. by real constants for multi-material elements (PIPE17, SOLID46, SOLID65, SHELL91, SHELL99, SOLID191)
3. by section data for multi-material elements (SHELL131, SHELL132, SHELL181, BEAM188, BEAM189, SHELL208, and SHELL209).

If a unique load is defined among merged nodes, the value is kept and applied to the retained node. If loads are not unique (not recommended), only the value on the lowest node (or highest if *Switch* = HIGH) will be kept, except for "force" loads for which the values will be summed if they are not defined using tabular boundary conditions.

Note — The unused nodes (not recommended) in elements, couplings, constraint equations, etc. may become active after the merge operation.

The *Action* field provides the option of visualizing the coincident items before the merging operation.

Caution: When merging entities in a model that has already been meshed, the order in which you issue multiple **NUMMRG** commands is significant. If you want to merge two adjacent meshed regions that have coincident nodes and keypoints, always merge nodes [**NUMMRG**,NODE] before merging keypoints [**NUMMRG**,KP]. Merging keypoints before nodes can result in some of the nodes becoming "orphaned"; that is, the nodes lose their association with the solid model. Orphaned nodes can cause certain operations (such as boundary condition transfers, surface load transfers, and so on) to fail.

After a **NUMMRG**,NODE, is issued, some nodes may be attached to more than one solid entity. As a result, subsequent attempts to transfer solid model loads to the elements may not be successful. Issue **NUMMRG**,KP to correct this problem. Do *NOT* issue **VCLEAR** before issuing **NUMMRG**,KP.

For **NUMMRG**,ELEM, elements must be identical in all aspects, including the direction of the element coordinate system.

When working with solid models, you may have better success with the gluing operations (**AGLUE**, **LGLUE**, **VGLUE**). Please read the following information when attempting to merge solid model entities.

Gluing Operations vs. Merging Operations

Adjacent, touching regions can be joined by gluing them (**AGLUE**, **LGLUE**, **VGLUE**) or by merging coincident keypoints (**NUMMRG**,KP, which also causes merging of identical lines, areas, and volumes). In many situations, either approach will work just fine. Some factors, however, may lead to a preference for one method over the other.

Geometric Configuration

Gluing is possible regardless of the initial alignment or offset of the input entities. Keypoint merging is possible only if each keypoint on one side of the face to be joined is matched by a coincident keypoint on the other side. This is commonly the case after a symmetry reflection (**ARSYM** or **VSYMM**) or a copy (**AGEN** or **VGEN**), especially for a model built entirely in ANSYS rather than imported from a CAD system. When the geometry is extremely

precise, and the configuration is correct for keypoint merging, **NUMMRG** is more efficient and robust than **AGLUE** or **VGLUE**.

Model Accuracy

Gluing, like all ANSYS boolean operations, requires that the input entities meet the current boolean tolerance (BTOL). Otherwise, **AGLUE** or **VGLUE** may fail. In such cases, relaxing the tolerance may allow the glue to complete. An advantage of gluing is that it is unlikely to degrade the accuracy of a geometric model. Keypoint merging can operate on almost any combination of entities (although you may have to override the default tolerances on **NUMMRG**). However, it can also introduce or increase accuracy flaws, making later boolean operations less likely to succeed. If the input tolerances are too large, **NUMMRG** can collapse out small lines, areas, or volumes you intended to keep, possibly rendering the model unusable.

Mesh Status

Gluing, like all ANSYS boolean operations, requires that the input entities be unmeshed. Keypoint merging is effective for meshed models under the right conditions. More information on keypoint merging follows.

Merging Solid Model Entities:

When merging solid model entities (*Label* = KP or ALL), keypoint locations are used as the basis for merging. Once keypoints are merged, any higher order solid model entities (lines, areas, and volumes) attached to those keypoints are considered for merging.

Keypoints that are attached to lines will be merged only if:

- ΔX, ΔY, and ΔZ are each less than *TOLER*

  where,

  ΔX is the X component of the distance between keypoints,
  ΔY is the Y component of the distance between keypoints, and
  ΔZ is the Z component of the distance between keypoints;

  and

- $\sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}$ is less than 1E-5 times the length of the longest line attached to those keypoints (internal relative solid model tolerance), or $\sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}$ is less than *GTOLER* (global solid model tolerance) if specified.

The *TOLER* field is a *consideration* tolerance. If a keypoint is within *TOLER* of another keypoint, then those two keypoints are *candidates* to be merged. If, when "moving" the higher numbered keypoint, the distance exceeds the internal relative solid model tolerance, or the global solid model tolerance (*GTOLER*) if specified, the keypoints will *not* be merged. Lines, areas, and volumes are considered for merging in a similar manner.

The internal relative solid model tolerance should be overridden by the global solid model tolerance (*GTOLER*) only when absolutely necessary. *GTOLER* is an *absolute* tolerance; if specified, relative lengths of lines in the model will no longer be considered in the merge operation. If *GTOLER* is too large, you can "merge-out" portions of your model accidently, effectively defeaturing the model. If *GTOLER* is used, it is recommended that **NUMMRG** be preceded by saving the database (since undesired merges of solid model entities could occur).

Note — Use of the **NUMMRG** command does not cause changes to a model's geometry; only the topology is affected.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Merge Nodes
Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts
Main Menu>Preprocessor>Numbering Ctrls>Merge Items

NUMOFF, *Label*, *VALUE*

Adds a number offset to defined items.

PREP7: Database

MP ME ST DY <> PR EM <> FL PP ED

Label

Apply offset number to one of the following sets of items:

NODE
Nodes

ELEM
Elements

KP
Keypoints

LINE
Lines

AREA
Areas

VOLU
Volumes

MAT
Materials

TYPE
Element types

REAL
Real constants

CP
Coupled sets

SECN
Section numbers

CE
Constraint equations

CSYS
Coordinate systems

VALUE
Offset number value (cannot be negative).

Notes

Useful for offsetting current model data to prevent overlap if another model is read in. **CDWRITE** automatically writes the appropriate **NUMOFF** commands followed by the model data to **File.CDB**. Therefore, when the file is read, any model already existing in the database is offset before the model data on the file is read.

Offsetting material numbers with this command [**NUMOFF**,MAT] does *not* update the material number referenced by any of the following:

- A temperature-dependent convection or surface-to-surface radiation load [**SF**, **SFE**, **SFL**, **SFA**]
- Real constants for multi-material elements (PIPE17, SOLID46, SOLID65, SHELL91, SHELL99, SOLID191)
- Section information for multi-material elements (SHELL131, SHELL132, SHELL181, BEAM188, BEAM189, SHELL208, and SHELL209). See also the **SECDATA** and **SECWRITE** command descriptions.

Therefore, a mismatch may exist between the material definitions and the material numbers referenced.

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Add Num Offset

NUMSTR, *Label*, *VALUE*

Establishes starting numbers for automatically numbered items.

PREP7: Database

MP ME ST DY <> PR EM <> FL PP ED

Label
Apply starting number to one of the following sets of items:

NODE
Node numbers. *Value* defaults (and is continually reset) to 1 + maximum node number in model. Cannot be reset lower.

ELEM
Element numbers. *Value* defaults (and is continually reset) to 1 + maximum element number in model. Cannot be reset lower.

KP
Keypoint numbers. *Value* defaults to 1. Only undefined numbers are used. Existing keypoints are not overwritten.

LINE
Line numbers. *Value* defaults to 1. Only undefined numbers are used. Existing lines are not overwritten.

AREA
Area numbers. *Value* defaults to 1. Only undefined numbers are used. Existing areas are not overwritten.

VOLU
Volume numbers. *Value* defaults to 1. Only undefined numbers are used. Existing volumes are not overwritten.

DEFA
Default. Returns all starting numbers to their default values.

VALUE
Starting number value.

Notes

Establishes starting numbers for various items that may have numbers automatically assigned (such as element numbers with the **EGEN** command, and node and solid model entity numbers with the mesh [**AMESH**, **VMESH**, etc.] commands). Use **NUMSTR**,STAT to display settings. Use **NUMSTR**,DEFA to reset all specifications back to defaults. Defaults may be lowered by deleting and compressing items (i.e., **NDELE** and **NUMCMP**,NODE for nodes, etc.).

Note — A mesh clear operation (**VCLEAR**, **ACLEAR**, **LCLEAR**, and **KCLEAR**) automatically sets starting node and element numbers to the highest unused numbers. If a specific starting node or element number is desired, issue **NUMSTR** after the clear operation.

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Reset Start Num
Main Menu>Preprocessor>Numbering Ctrls>Set Start Number
Main Menu>Preprocessor>Numbering Ctrls>Start Num Status

NUMVAR, *NV*

Specifies the number of variables allowed in POST26.

POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

NV
Allow storage for *NV* variables. 200 maximum are allowed. Defaults to 10 (except for an explicit dynamics analysis, which defaults to 30). TIME (variable 1) should also be included in this number.

Command Default

Allow storage for 10 variables (30 variables for an explicit dynamics analysis).

Notes

Specifies the number of variables allowed for data read from the results file and for data resulting from an operation (if any). For efficiency, *NV* should not be larger than necessary. *NV* cannot be changed after data storage begins.

Menu Paths

Main Menu>TimeHist Postpro>Settings>File

NUSORT

Restores original order for nodal data.

POST1: Results

MP ME ST DY <> PR EM <> FL PP ED

Notes

This command restores the nodal data to its original order (sorted in ascending node number sequence) after an **NSORT** command. Changing the selected nodal set [**NSEL**] also restores the original nodal order.

Menu Paths

Main Menu>General Postproc>List Results>Sorted Listing>Unsort Nodes

---

NWPAVE, *N1*, *N2*, *N3*, *N4*, *N5*, *N6*, *N7*, *N8*, *N9*

Moves the working plane origin to the average location of nodes.

DATABASE: Working Plane

MP ME ST DY <> PR EM <> FL PP ED

N1, *N2*, *N3*, *N4*, *N5*, *N6*, *N7*, *N8*, *N9*

Nodes used in calculation of the average. At least one must be defined. If *N1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Averaging is based on the active coordinate system.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Offset WP to>Nodes

NWPLAN, *WN*, *NORIG*, *NXAX*, *NPLAN*

Defines the working plane using three nodes.

DATABASE: Working Plane

MP ME ST DY <> PR EM <> FL PP ED

WN

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified. If fewer than three points are used, the viewing direction of window *WN* will be used instead to define the normal to the working plane.

NORIG

Node number defining the origin of the working plane coordinate system. If *NORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NXAX
Node number defining the x-axis orientation (defaults to the x-axis being parallel to the global X-axis; or if the normal to the working plane is parallel to the global X-axis, then defaults to being parallel to the global Y-axis).

NPLAN
Node number defining the working plane (the normal defaults to the present display view [**/VIEW**] of window *WN*).

Notes

Defines a working plane to assist in picking operations using three nodes as an alternate to the **WPLANE** command. The three nodes also define the working plane coordinate system. A minimum of one node (at the working plane origin) is required. Immediate mode may also be active. See the **WPSTYL** command to set the style of the working plane display.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node
Utility Menu>WorkPlane>Align WP with>Nodes

NWRITE, *Fname*, *Ext*, --, *KAPPND*

Writes nodes to a file.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to NODE if *Fname* is blank.

--
Unused field

KAPPND
Append key:

0
Rewind file before the write operation.

1
Append data to the end of the existing file.

Notes

Writes selected nodes [**NSEL**] to a file. The write operation is not necessary in a standard ANSYS run but is provided as a convenience to users wanting a coded node file. Data are written in a coded format. The format used is (I8, 6G20.13) to write out *NODE*,*X*,*Y*,*Z*,*THXY*,*THYZ*,*THZX*. If the last number is zero (i.e., *THZX* = 0), or the last set of numbers are zero, they are not written but are left blank. Therefore, you must use a formatted read to process this file. Coordinate values are in the global Cartesian system.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>Write Node File

O Commands

OMEGA, *OMEGX*, *OMEGY*, *OMEGZ*, *KSPIN*
Specifies the rotational velocity of the structure.

SOLUTION: Inertia

MP ME ST <> <> PR <> <> FL PP ED

OMEGX, *OMEGY*, *OMEGZ*
Rotational velocity of the structure about the global Cartesian X, Y, and Z axes.

KSPIN
Spin softening key:

0
No modification of stiffness matrix due to rotation.

1
Decrease radial stiffness due to rotation (i.e., include spin softening effects).

Notes

Specifies the rotational velocity of the structure about each of the global Cartesian axes (right-hand rule). Rotational velocities may be defined in analysis types **ANTYPE**,STATIC, HARMIC (full or mode superposition), TRANS (full or mode superposition), and SUBSTR. Rotational velocities are combined with the element mass matrices to form a body force load vector term. Units are radians/time. Related commands are **ACEL**, **CGLOC**, **CGOMGA**, **DCGOMG**, and **DOMEGA**.

The *KSPIN* option allows adjusting the stiffness of a rotating body to account for dynamic mass effects. The adjustment is called *spin-softening* and applies to a modal (**ANTYPE**,MODAL) or harmonic (**ANTYPE**,HARM) analysis only. The adjustment approximates the effects of geometry changes caused by large-deflection circumferential motion in a small-deflection analysis. The *KSPIN* option is not intended for a large-deflection static analysis; in such a case, use the **NLGEOM**,ON command to account for this effect. The command is usually used in conjunction with prestressing (**PSTRES**).

The overall stiffness matrix should not be reused (**KUSE**) if **OMEGA** command parameters change between substeps. See the *ANSYS, Inc. Theory Reference* for matrix details.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>Global
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Veloc>Global
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>Global
Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Veloc>Global

OPADD, *NRES*, *NUM1*, *NUM2*, *C1*, *C2*

Forms a set of optimization parameters by adding two sets.

OPTIMIZATION: Operations

MP ME ST DY <> PR EM <> FL PP ED

NRES
Number assigned to results set. If same as existing set, the existing values will be overwritten by these results (defaults to next available set number).

NUM1
Number of first design set to be used in operation.

NUM2
Number of second design set to be used in operation (may be blank).

C1
Scale factor applied to *NUM1* (defaults to 1.0).

C2
Scale factor applied to *NUM2* (defaults to 1.0).

Notes

Forms a result set of parameters by adding two existing design sets according to the operation:

$$NRES = (C1 \times NUM1) + (C2 \times NUM2)$$

OPADD can also be used to scale results for a single set. If no set number is specified for the results set (*NRES*), it defaults to the next available set number. Because no compression of set numbers is done, the next available set number will be either 1 + the highest existing set number, or 1 + the highest set number used (if sets with higher numbers previously existed but have been removed).

Menu Paths

Main Menu>Design Opt>Design Sets>Combine

OPANL, *Fname*, *Ext*, --

Defines the analysis file to be used for optimization looping.

OPTIMIZATION: Files

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Ext
Filename extension (8 character maximum).

--
Unused field

Command Default

If interactive, no default; if batch, use the batch input stream (**File.BAT**).

Notes

The optimization looping file must be specified for an optimization which is performed interactively. The file must exist at the time **OPANL** is issued. In this file, where the **/PREP7** and **/OPT**commands occur, they must be the first nonblank characters on the line (i.e., do not use the $ delimiter on any **/PREP7** or **/OPT** command lines).

Menu Paths

Main Menu>Design Opt>Analysis File>Assign

OPCLR

Clears the optimization database.

OPTIMIZATION: Operations

MP ME ST DY <> PR EM <> FL PP ED

Notes

Clears the optimization database. All settings are reset to their default values, and all design sets are deleted. This command is useful between multiple (independent) optimization analyses in the same batch run or interactive session.

Menu Paths

Main Menu>Design Opt>Opt Database>Clear & Reset

OPDATA, *Fname*, *Ext*, --

Identifies the file where optimization data is to be saved.

OPTIMIZATION: Files

MP ME ST DY <> PR EM <> FL PP ED

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext

Filename extension (8 character maximum).

The extension defaults to OPT if *Fname* is blank.

--

Unused field

Command Default

Optimization data is saved on **File.OPT**.

Notes

Identifies the file where optimization data is to be automatically saved during looping. Automatic saves occur at the end of each loop or iteration, and upon finishing out of the optimizer [**FINISH**]. (To save the data elsewhere in the optimizer, use **OPSAVE**.) Optimization data written to the file include DVs, SVs, saved design sets, analysis file name, etc. The data can be restored via the **OPRESU** command.

Menu Paths

Main Menu>Design Opt>Controls

OPDEL, *NSET1*, *NSET2*

Deletes optimization design sets.

OPTIMIZATION: Operations

MP ME ST DY <> PR EM <> FL PP ED

NSET1

First design set in range to be deleted. Defaults to 0. If *NSET1* is blank and *NSET2* is ≥ 1, all sets from the lowest existing set number to *NSET2* will be deleted. If *NSET1* = ALL, all design sets are removed.

NSET2

Last design set in range to be deleted. Defaults to *NSET1*.

Notes

Deletes the design sets in a specified range. All sets occurring within that range (i.e., $NSET1 \leq N \leq NSET2$, where N is an existing set number) are permanently removed from the optimization database. The original set numbers are retained for remaining design sets. Note that no compression of set numbers is done. Thus, if the highest set is deleted, the next available set number will be the previously existing highest set number + 1. (A maximum of 130 design sets can be stored in the optimization database.)

Menu Paths

Main Menu>Design Opt>Design Sets>Select/Delete

OPEQN, *KFOBJ*, *KFSV*, *KWGHT*, *KOPPR*, *INOPT*

Controls curve fitting for the subproblem approximation method.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

KF0BJ

Objective function curve fit:

0
Quadratic plus cross-term curve fit for objective function (default).

1
Linear curve fit.

2
Quadratic curve fit.

3
Quadratic plus cross-term fit (same as 0).

KFSV
State variables curve fit:

0
Quadratic curve fit for state variables (default).

1
Linear curve fit.

2
Quadratic curve fit (same as 0).

3
Quadratic plus cross-term curve fit.

KWGHT
Weighting factors:

0
Weights applied to design sets based on triple products of distance in design space, objective function values, and feasibility/infeasibility (*KWGHT* = 2, 3, and 4, below) (default).

1
All weights set to unity.

2
Weights based on distance in design space.

3
Weights based on objective function values.

4
Weights based on feasibility/infeasibility.

KOPPR
Optimization printout:

0
No printout of approximation data (default).

N
Printout every *n*th optimization loop.

INOPT
Approximation reformulation:

0
Approximations are updated every loop (default).

N
Approximations are completely reformulated every *n*th loop.

Notes

Controls the curve fitting for the optimization approximations used with the subproblem approximation method [**OPTYPE**,SUBP]. See the *ANSYS, Inc. Theory Reference* for details.

Menu Paths

Main Menu>Design Opt>Method/Tool

OPERATE

Specifies "Operation data" as the subsequent status topic.

POST26: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

This command cannot be accessed from a menu.

OPEXE

Initiates optimization looping.

OPTIMIZATION: Run

MP ME ST DY <> PR EM <> FL PP ED

Notes

Initiates optimization looping using commands condensed from the analysis file [**OPANL**]. The type of optimization to be performed is specified by the **OPTYPE** command. Upon issuing this command, optimization looping begins and the design variables are updated as necessary based on the type of optimization chosen. Control remains with the ANSYS program until convergence or termination occurs, at which time the commands occurring after **OPEXE** are executed. This command is not allowed within a do-loop [***DO**].

Menu Paths

Main Menu>Design Opt>Run

OPFACT, *Type*

Defines the type of factorial evaluation to be performed.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

Type

Type of factorial evaluation. Number of iterations is determined by the number of design variables (*n*).

FULL

Perform a full factorial analysis (default); 2**n iterations are required. n must be $\leq$ 7.

1/2

Perform a 1/2 fractional factorial analysis. (1/2)*2**n iterations are required. n must be $\leq$ 8.

1/4

Perform a 1/4 fractional factorial analysis. (1/4)*2**n iterations are required. n must be $\leq$ 9.

1/8

Perform a 1/8 fractional factorial analysis. (1/8)*2**n iterations are required. n must be $\leq$ 10.

1/16

Perform a 1/16 fractional factorial analysis. (1/16)*2**n iterations are required. n must be $\leq$ 10.

1/32

Perform a 1/32 fractional factorial analysis. (1/32)*2**n iterations are required. n must be $\leq$ 10.

1/64

Perform a 1/64 fractional factorial analysis. (1/64)*2**n iterations are required. n must be $\leq$ 10.

Command Default

Perform a full factorial analysis.

Notes

This command is valid for the factorial evaluation method of optimization [**OPTYPE**,FACT]. The factorial evaluation is defined as either full or fractional. The number of iterations performed is dependent on the number of design variables that have been defined (*n*).

Menu Paths

Main Menu>Design Opt>Method/Tool

OPFRST, *NITR*, *SIZE*, *DELTA*

Defines specifications for the first order optimization method.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

NITR

Maximum number of optimization iterations for the next execution [**OPEXE**]. Defaults to previously specified value, if any, otherwise defaults to 10.

SIZE

Limit (in percent) that is applied to the size of each line search step. For example, setting *SIZE* = 10 will limit design variable changes to 10 percent of the maximum range of design space (defined by feasible design variable limits specified on **OPVAR** commands) at each iteration. Defaults to previously specified value, if any; otherwise, defaults to 100 (percent).

DELTA

The forward difference (in percent) applied to the design variable range that is used to compute the gradient. For example, the shift in a DV is *DELTA**(MAX - MIN)/100, where MIN and MAX are specified on the **OPVAR** command. Defaults to previously specified value, if any; otherwise, defaults to 0.2 (percent).

Command Default

NITR = 10, *SIZE* = 100, *DELTA* = 0.2.

Notes

This command is valid for the first order method of optimization [**OPTYPE**,FIRST].

Menu Paths

Main Menu>Design Opt>Method/Tool

OPGRAD, *Dset*, *DELTA*

Specifies which design set will be used for gradient evaluation.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

Dset

BEST
Use the best design set as the evaluation point (default).

LAST
Use the last design set as the evaluation point.

n
Use design set number *n* as the evaluation point.

DELTA
The forward difference applied to the design variable range that is used to compute the gradient. For example, the shift in a design variable (DV) is defined as *DELTA**(MAX-MIN)/100, with MIN and MAX as specified on the **OPVAR** command. Defaults to 0.5.

Command Default

As described for each argument above.

Notes

This command is valid for the gradient evaluation method of optimization [**OPTYPE**,GRAD]. **OPGRAD** specifies which point (design set) in design space will be used to evaluate the gradient and what the forward difference will be.

Menu Paths

Main Menu>Design Opt>Method/Tool

OPKEEP, *Key*

Specifies whether to save the best-set results and database file.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

Key
Save key:

OFF
Do not save the results and database files for the best design set (default).

ON
Save results and database files (**File.BRST** and **File.BDB**) for the best design set during optimization looping.

Command Default

Do not save the database and results for the best design set.

Notes

Specifies whether or not the results file and database file corresponding to the best design set (based on all existing sets) should be saved during optimization. The saved files will be named **File.BRST** (or **File.BRTH** for thermal analysis, **File.BRMG** for magnetic analysis, **File.BRFL** for a FLOTRAN CFD analysis) and **File.BDB**. If *Key* = ON, both files will be saved at the end of each optimization loop which results in a new best design set. **File.RST** (results) and **File.DB** (database) will be saved as usual for the last design set, regardless of the *Key* setting. If multiple analyses are done within an optimization loop, then only the last analysis is considered for the *Key* = ON option. For example, if a thermal analysis is followed by a stress analysis, the files saved for the best set will be **File.BRST** and **File.BDB**, which will both correspond to the structural analysis.

Menu Paths

Main Menu>Design Opt>Controls

OPLFA, *Name*, *Effect*, *EMIN*, *EMAX*

Displays the results of a factorial evaluation.

OPTIMIZATION: Display

MP ME ST DY <> PR EM <> FL PP ED

Name
Parameter name. The parameter must have been previously defined as the objective function or a state variable [**OPVAR**].

Effect
Display effects for parameter *Name*.

MAIN
Display main effects (default).

2FAC
Display two-factor interactions.

3FAC
Display three-factor interactions.

EMIN
Minimum value of effect to be displayed. Default - computed minimum.

EMAX
Maximum value of effect to be displayed. Default - computed maximum.

Notes

Displays the results of a factorial analysis in bar chart form. Only the 10 largest interaction values will be displayed in the specified range (*EMIN* to *EMAX*). This command is only valid after a factorial evaluation [**OPTYPE**,FACT].

Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Graph>Factorial

---

OPLGR, *Pname*, *Dvnam1*, *Dvnam2*, *Dvnam3*, *Dvnam4*, *Dvnam5*, *Dvnam6*

Graphs the results of a gradient evaluation.

OPTIMIZATION: Display

MP ME ST DY <> PR EM <> FL PP ED

Pname
Response parameter name. The parameter must have been previously defined as the objective function or a state variable [**OPVAR**].

Dvnam1, *Dvnam2*, *Dvnam3*, *Dvnam4*, *Dvnam5*, *Dvnam6*
Design variable names.

Notes

This command is only valid after a gradient evaluation [**OPTYPE**,GRAD]. *Pname* is graphed with respect to a plus or minus 1% change in design variable (*Dvnam1*, ... *Dvnam6*).

Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Graph>Gradient

OPLIST, *SET1*, *SET2*, *LKEY*

Displays the parameters for design sets.

OPTIMIZATION: Display

MP ME ST DY <> PR EM <> FL PP ED

SET1, *SET2*
Display values of all analysis parameters from *SET1* (defaults to the highest existing set) to *SET2* (defaults to *SET1*). If *SET1* = ALL, display for all sets.

LKEY
Listing key:

0
List only scalar parameters related to optimization.

1
List all analysis scalar parameters.

Notes

Displays the values of the parameters for specified design sets. Note that a ***STATUS** command may also be issued to display various optimization data.

Menu Paths

Main Menu>Design Opt>Design Sets>List

OPLOOP, *Read*, *Dvar*, *Parms*

Specifies controls for optimization looping.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

Read
Indicates where to begin reading the analysis file during optimization looping. Note that the **/PREP7** command (and the **/OPT** command) must occur as the first nonblank characters on a line in the analysis file (i.e., do not use the $ delimiter). Defaults to previously specified label, if any; otherwise, defaults to TOP:

TOP
Read from the first line.

PREP
Read from the first occurrence of **/PREP7**.

Dvar
Indicates how to treat parameters which have been designated as design variables (DVs) during optimization looping. Defaults to previously specified label, if any; otherwise, defaults to IGNORE:

IGNORE
Do not process DV parameter assignments in the analysis file during looping.

PROCESS
Process DV parameter assignments in the analysis file during looping.

Parms
Indicates which types of parameters to save during optimization looping. Defaults to previously specified label, if any; otherwise, defaults to SCALAR:

SCALAR
Save scalar parameters only.

ALL
Save all parameters, both scalar and array.

Command Default

Read = TOP, *Dvar* = IGNORE, and *Parms* = SCALAR.

Menu Paths

Main Menu>Design Opt>Controls

OPLSW, *Pname*, *Dvnam1*, *Dvnam2*, *Dvnam3*, *Dvnam4*, *Dvnam5*, *Dvnam6*
Graphs the results of a global sweep generation.

OPTIMIZATION: Display

MP ME ST DY <> PR EM <> FL PP ED

Pname
Response parameter name. The parameter must have been previously defined as the objective function or a state variable (**OPVAR**).

Dvnam1, *Dvnam2*, *Dvnam3*, *Dvnam4*, *Dvnam5*, *Dvnam6*
Design variable names.

Notes

This command is only valid after a global sweep generation [**OPTYPE**,SWEEP]. *Pname* is graphed with respect to the specified design variables (*Dvnam1*, ... *Dvnam6*). The design variables are normalized (0 to 1) along the X-axis.

Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Graph>Sweeps

OPMAKE

Creates a design set using active scalar parameter values.

OPTIMIZATION: Operations

MP ME ST DY <> PR EM <> FL PP ED

Notes

The next available design set number is assigned to the new design set. Because no compression of set numbers is done, the next available set number will be either 1 + the highest existing set number, or 1 + the highest set number used (if sets with higher numbers previously existed but have been removed). Multiple design sets can be created by repeated use of **PARRES** (to read parameters from a file) and **OPMAKE**.

Menu Paths

Main Menu>Design Opt>Design Sets>Create

OPNCONTROL, *Lab*, *VALUE*, *NUMSTEP*

Sets decision parameter for automatically increasing the time step interval.

SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> <> PP ED

Lab

DOF label used to base a decision for increasing the time step (substep) interval in a nonlinear or transient analysis. The only DOF label currently supported is TEMP.

VALUE, *NUMSTEP*

Two values used in algorithm for deciding if the time step interval can be increased. The time step interval is increased if the maximum absolute value of the incremental solution at the specified DOF label is less than *VALUE* for the number of contiguous time steps specified by *NUMSTEP*. The default for *VALUE* is 0.1 and the default for *NUMSTEP* is 3.

Notes

This command is available only for nonlinear or transient analysis, and only when **SOLCONTROL**,ON.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Open Control
Main Menu>Solution>Load Step Opts>Nonlinear>Open Control

OPPRNT, *Key*

Activates detailed optimization summary printout.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

Key
Summary key:

OFF
Do not print details (default).

ON
Print the detailed summary.

FULL
Same as *Key* = ON, but includes a full list of all design sets.

Notes

Activates detailed optimization summary printout to the primary output during optimization analysis.

Menu Paths

Main Menu>Design Opt>Controls

OPRAND, *NITR*, *NFEAS*

Defines the number of iterations for a random optimization.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

NITR
Maximum number of optimization iterations for the next execution [**OPEXE**]. Defaults to previously specified value, if any; otherwise, defaults to 1.

NFEAS
Desired total number of feasible design sets (including existing feasible sets). Once *NFEAS* feasible sets are achieved, random iterations are terminated. If 0 is input, *NITR* random iterations will be performed independent of the number of feasible designs that are obtained. Defaults to previously specified value, if any; otherwise, defaults to 0.

Command Default

Perform 1 iteration.

Notes

This command is valid for the random design generation method of optimization [**OPTYPE**,RAND].

Menu Paths

Main Menu>Design Opt>Method/Tool

OPRESU, *Fname*, *Ext*, --

Reads optimization data into the optimization database.

OPTIMIZATION: Files

MP ME ST DY <> PR EM <> FL PP ED

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext

Filename extension (8 character maximum).

The extension defaults to OPT if *Fname* is blank.

--

Unused field

Notes

Reads optimization data from the specified file into the optimization database. Optimization data must have been previously saved to this file [**OPSAVE**], or automatically written to this file in an earlier optimization analysis. All existing parameters will be deleted and replaced by those stored on the resumed file. (To save existing parameters, use **PARSAV** before **OPRESU**.)

Menu Paths

Main Menu>Design Opt>Opt Database>Resume

OPRFA, *Name*

Prints the results of a factorial evaluation.

OPTIMIZATION: Display

MP ME ST DY <> PR EM <> FL PP ED

Name

Parameter name. The parameter must have been previously defined as the objective function or a state variable [**OPVAR**]. If *Name* = ALL, print results for all optimization response variables (default).

Notes

This command is only valid after a factorial evaluation [**OPTYPE**,FACT].

Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Print

OPRGR, *Name*

Prints the results of a gradient evaluation.

OPTIMIZATION: Display

MP ME ST DY <> PR EM <> FL PP ED

Name

Parameter name. The parameter must have been previously defined as the objective function or a state variable [**OPVAR**]. If *Name* = ALL, print results for all optimization response variables (default).

Notes

This command is only valid after a gradient evaluation [**OPTYPE**,GRAD].

Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Print

OPRSW, *Name*

Prints the results of a global sweep generation.

OPTIMIZATION: Display

MP ME ST DY <> PR EM <> FL PP ED

Name

Parameter name. The parameter must have been previously defined as the objective function or a state variable (**OPVAR**). If *Name* = ALL, print results for all optimization response variables (default).

Notes

This command is only valid after a global sweep generation [**OPTYPE**,SWEEP].

Menu Paths

Main Menu>Design Opt>Design Sets>Tool Results>Print

OPSAVE, *Fname*, *Ext*, --

Writes all optimization data to a file.

OPTIMIZATION: Files

MP ME ST DY <> PR EM <> FL PP ED

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext

Filename extension (8 character maximum).

The extension defaults to OPT if *Fname* is blank.

--

Unused field

Notes

Writes all optimization data to **File.OPT** or a named file. Saved data includes optimization data only (DVs, SVs, saved design sets, analysis file name, etc.). The data can be restored [**OPRESU**] for an optimization restart.

Menu Paths

Main Menu>Design Opt>Opt Database>Save

OPSEL, *NSEL*

Selects design sets for subsequent optimization looping.

OPTIMIZATION: Operations

MP ME ST DY <> PR EM <> FL PP ED

NSEL

Number of best design sets to be selected. If *NSEL* = -1, select all feasible designs. If *NSEL* is positive and no objective function is defined, the following occurs: all infeasible design sets are removed; next the lowest number (oldest) feasible design sets are removed until *NSEL* sets are left (i.e., the latest feasible design sets are kept).

Notes

A number of best design sets or all feasible design sets may be specified. ("Best" design sets are determined by the objective function value for each feasible design.) All design sets not selected are permanently removed from the optimization database. The original set numbers are retained for remaining design sets. Note that no compression of set numbers is done. Thus, if the highest set is removed, the next available set will still be the previously used highest set number + 1. (A maximum of 130 design sets can be stored in the optimization database.)

Menu Paths

Main Menu>Design Opt>Design Sets>Select/Delete

OPSUBP, *NITR*, *NINFS*

Defines number of iterations for subproblem approximation method.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

NITR

Maximum number of optimization iterations for the next execution [**OPEXE**]. Defaults to previously specified value, if any; otherwise, defaults to 30.

NINFS

Maximum number of consecutive infeasible design sets that are allowed. Once *NINFS* consecutive infeasible sets are encountered, iterations are terminated. Defaults to previously specified value, if any; otherwise, defaults to 7.

Command Default

NITR = 30, *NINFS* = 7.

Notes

This command is valid for the subproblem approximation method of optimization [**OPTYPE**,SUBP].

Menu Paths

Main Menu>Design Opt>Method/Tool

OPSWEEP, *Dset*, *NSPS*

Specifies the reference point and number of evaluation points for a sweep generation.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

Dset

BEST
Use the best design set as the reference point (default).

LAST
Use the last design set as the reference point.

N
Use design set number N as the reference point.

NSPS

Number of evaluation points used for each design variable sweep. Two to ten evaluation points are allowed. Defaults to 2 (i.e., the minimum and maximum design variable values).

Command Default

As described by the arguments above.

Notes

This command is valid for the sweep evaluation method of optimization [**OPTYPE**,SWEEP].

Menu Paths

Main Menu>Design Opt>Method/Tool

/OPT

Enters the design optimizer.

SESSION: Processor Entry
OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

Notes

Enters the ANSYS optimizer for design optimization by iterative analyses. Upon issuing this command, if there are no existing design sets, all currently defined numerical scalar parameters are established as design set number 1.

This command is valid only at the Begin Level.

Menu Paths

Main Menu>Design Opt

OPTYPE, *Mname*

Specifies the optimization method to be used.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM EH FL PP ED

Mname
Name of the optimization method to be used:

SUBP
Subproblem approximation method. Specifications for this method are defined on the **OPSUBP** command.

FIRST
First order optimization method. Specifications for this method are defined on the **OPFRST** command.

RAND
Perform random iterations. Specifications for this method are defined on the **OPRAND** command.

RUN
Perform a single optimization run with current parameter values for the defined design variables.

FACT
Perform factored design iterations. Specifications for this method are defined on the **OPFACT** command.

GRAD
Compute a gradient (design sensitivity) at a point in design space. Specifications for this method are defined on the **OPGRAD** command.

SWEEP
Perform global sweeps through global design space starting from a single design set. Specifications for this method are defined on the **OPSWEEP** command.

USER
User-supplied external optimization. See the *ANSYS Advanced Analysis Techniques Guide* for user optimization information (USEROP routine). Parameters used for this method are defined on the **OPUSER** command.

Command Default

Perform a single optimization run (*Mname* = RUN) with current parameter values for the defined design variables.

Notes

Specifies the optimization method to be used upon execution [**OPEXE**].

Product Restrictions

Mname = USER is only allowed in ANSYS Multiphysics, ANSYS Mechanical, ANSYS Structural, and ANSYS PrepPost.

Menu Paths

Main Menu>Design Opt>Method/Tool

OPUSER, *NITR*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*
Defines specifications for user-supplied external optimization.

OPTIMIZATION: Specifications

MP ME ST <> <> <> <> <> <> PP <>

NITR
Maximum number of optimization iterations for the next execution [**OPEXE**]. Defaults to previously specified value, if any; otherwise, defaults to 1.

VAL1, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*
Numerical *values* (double precision) for user-supplied optimizer. Values may be the results of parameter evaluations. Values are written to **File.OPT** after the parameter set data. *VAL1*--*VAL8* default to previously specified values, if any; otherwise, default to 0.

Command Default

NITR = 1, *VAL1* through *VAL8* equal zero.

Notes

Defines the number of iterations and the parameter values for user-supplied external optimization [**OPTYPE**,USER]. Values are passed to the user routine (USEROP) through the optimization common block.

Menu Paths

Main Menu>Design Opt>Method/Tool

OPVAR, *Name*, *Type*, *MIN*, *MAX*, *TOLER*

Specifies the parameters to be treated as optimization variables.

OPTIMIZATION: Specifications

MP ME ST DY <> PR EM <> FL PP ED

Name
Parameter name (must be a scalar parameter).

Type
Optimization variable type:

DV
Design variable. *MAX* must also be specified.

SV
State variable, also referred to as a constrained variable. (Usually defined via a ***GET** command.) *MIN*, or *MAX*, or both must be specified.

OBJ
Objective function (variable to be minimized). Only one objective function is allowed. *MIN* and *MAX* are not used.

DEL
Deletes this optimization variable (does not delete the parameter). This option only has meaning if the parameter *Name* was previously defined as an optimization variable (DV, SV, or OBJ). The parameter retains the value assigned during the last loop. *MIN* and *MAX* are not used.

MIN
Minimum value for this parameter. For *Type* = DV, *MIN* must be greater than 0.0 and defaults to 0.001*(*MAX*). For *Type* = SV, there will be no lower limit if *MIN* is left blank, but if *MIN* = 0.0, the lower limit is zero. *MIN* is ignored for *Type* = OBJ. *MIN* is also ignored for *Type* = DEL (except for the special case when *Name* = (blank) and *Type* = DEL, as described next).

When the GUI is on, if a **Delete** operation (in a **Design Variables** or **State Variables** dialog box) writes this command to a log file (**Jobname.LOG** or **Jobname.LGW**), you will observe that *Name* is blank, *Type* = DEL, and *MIN* is an integer number. In this case, the GUI has assigned a value of *MIN* that corresponds to the location of a chosen variable name in the dialog box's list. It is *not* intended that you type in such a location value for *MIN* in an ANSYS session. However, a file that contains a GUI-generated **OPVAR** command of this form can be used for batch input or for use with the **/INPUT** command.

MAX
Maximum value for this parameter. Required for *Type* = DV and ignored for *Type* = OBJ or DEL. For *Type* = SV, there will be no upper limit if *MAX* is left blank, but if *MAX* = 0.0, the upper limit is zero.

TOLER
For *Type* = DV and OBJ, acceptable change between loops for convergence; defaults to 0.01*(current value) for OBJ and 0.01*(*MAX*-*MIN*) for DV. For *Type* = SV, defines feasibility tolerance. For double-sided SV limits (*MAX* and *MIN* specified), the default is 0.01*(*MAX*-*MIN*). For a single-sided SV limit (*MAX* or *MIN* specified), the default is 0.01*(specified limit) or, if the absolute value of the limit is < 1, the default is 0.01*(current SV value). *TOLER* is ignored for *Type* = DEL.

Command Default

No optimization parameters defined.

Notes

Specifies the parameters to be treated as optimization variables. Variable constraints (*MIN* and *MAX*) and tolerances (*TOLER*) are also specified on this command. If the specified parameter (*Name*) does not exist at the time **OPVAR** is issued, the **OPVAR** command defines that parameter and assigns a value of zero to it. Up to 60 DV and 100 SV parameters are allowed.

Menu Paths

Main Menu>Design Opt>Design Variables
Main Menu>Design Opt>Objective
Main Menu>Design Opt>State Variables

OUTOPT

Specifies "Output options" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Output Options

OUTPR, *Item*, *FREQ*, *Cname*

Controls the solution printout.

SOLUTION: Misc Loads

MP ME ST <> <> PR EM <> FL PP ED

Item
Item for print control:

BASIC
Basic quantities (nodal DOF solution, nodal reaction loads, and element solution) (default).

NSOL
Nodal DOF solution.

RSOL
Nodal reaction loads.

ESOL
Element solution.

NLOAD
Element nodal loads.

VENG
Element energies.

V
Nodal velocity (applicable to structural transient analysis only (**ANTYPE**,TRANS)).

A
Nodal acceleration (applicable to structural transient analysis only (**ANTYPE**,TRANS)).

ALL
All of the above solution items.

FREQ
Print solution for this item every *FREQ*th (and the last) substep of each load step. If *-n*, print up to *n* equally spaced solutions (for automatic time stepping). If NONE, suppress all printout for this item for this load step. If ALL, print solution for this item for every substep. If LAST, print solution for this item only for the last substep of each load step. For a modal analysis, use NONE or ALL.

Cname
Name of the component, created with the **CM** command, defining the selected set of nodes or elements for which this specification is active. If blank, the set is all entities.

Note — The component named must be of the same type as the item, i.e. nodal or element. A component name is not allowed with the BASIC or ALL labels.

Command Default

No printout.

Notes

Controls the solution items to be printed, the frequency with which they are printed (in static, transient, or full harmonic analyses), and the set of nodes or elements to which this specification applies. An item is associated

with either a node (**NSOL**, **RFORCE**, V, and A items) or an element (all of the remaining items). The specifications are processed in the order that they are input. Up to 50 specifications (**OUTPR** and **OUTRES** commands combined) may be defined. Use **OUTPR**,STAT to list the current specifications and use **OUTPR**,ERASE to erase all the current specifications.

As described above, **OUTPR** writes some or all items (depending on analysis type) for all elements. To restrict the solution printout, use **OUTPR** to selectively suppress (*FREQ* = NONE) the writing of solution data, or first suppress the writing of all solution data (**OUTPR**,ALL,NONE) and then selectively turn on the writing of solution data with subsequent **OUTPR** commands.

If the generalized plane strain feature is active and **OUTPR** is issued, the change of fiber length at the ending point during deformation and the rotation of the ending plane about X and Y during deformation will be printed if any displacement at the nodes is printed. The reaction forces at the ending point will be printed if any reaction force at the nodes is printed.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Solu Printout
Main Menu>Solution>Load Step Opts>Output Ctrls>Solu Printout

/OUTPUT, *Fname*, *Ext*, --, *Loc*

Redirects text output to a file or to the screen.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Fname
Filename and directory path (248 character maximum, including directory) to which text output will be redirected (defaults to **Jobname** if *Ext* is specified). For interactive runs, *Fname* = TERM (or blank) redirects output to the screen. For batch runs, *Fname* = blank (with all remaining command arguments blank) redirects output to the default system output file.

Ext
Filename extension (8 character maximum).

--
Unused field

Loc
Location within a file to which output will be written:

(blank)
Output is written starting at the top of the file (default).

APPEND
Output is appended to the existing file.

Command Default

Text output is written to the screen for interactive runs and to the system output file for batch runs (see the *ANSYS Basic Analysis Guide*).

Notes

Text output includes responses to every command and GUI function, notes, warnings, errors, and other informational messages. Upon execution of **/OUTPUT**,*Fname*, *Ext*, . . . , all subsequent text output is redirected to the file ***Fname*.*Ext***. To redirect output back to the default location, issue **/OUTPUT** (no arguments).

Note — When using the GUI, output from list operations [**NLIST**, **DLIST**, etc.] is always sent to a list window regardless of the **/OUTPUT** setting. The output can then be saved on a file or copied to the **/OUTPUT** location using the *File* menu in the list window.

This command is valid in any processor.

Menu Paths

Utility Menu>File>Switch Output to>File
Utility Menu>File>Switch Output to>Output Window

OUTRES, *Item*, *FREQ*, *Cname*

Controls the solution data written to the database.

SOLUTION: Misc Loads

MP ME ST <> <> PR EM <> <> PP ED

Item
Results item for database and file write control:

ALL
All solution items except SVAR and LOCI records. This value is the default.

ERASE
Resets **OUTRES** specifications to their default values.

STAT
Lists the current **OUTRES** specifications.

BASIC
Write only NSOL, RSOL, NLOAD, STRS, FGRAD, and FFLUX records to the results file and database.

NSOL
Nodal DOF solution.

RSOL
Nodal reaction loads.

V
Nodal velocity (applicable to structural transient analysis only (**ANTYPE**,TRANS)).

A
Nodal acceleration (applicable to structural transient analysis only (**ANTYPE**,TRANS)).

ESOL
Element solution (includes all items following):

NLOAD
Element nodal, input constraint, and force loads (also used with the **/POST1** commands **PRRFOR**, **NFORCE**, and **FSUM** to calculate reaction loads).

STRS
Element nodal stresses.

EPEL
Element elastic strains.

EPTH
Element thermal, initial, and swelling strains.

EPPL
Element plastic strains.

EPCR
Element creep strains.

FGRAD
Element nodal gradients.

FFLUX
Element nodal fluxes.

LOCI
Integration point locations.

SVAR
State variables (used only by USERMAT).

MISC
Element miscellaneous data (SMISC and NMISC items of the **ETABLE** command).

FREQ
Specifies how often (that is, at which substeps) to write the specified solution results item. The following values are valid:

Value	Description
n	Writes the specified results item every *n*th (and the last) substep of each load step.
-*n*	Writes up to *n* equally spaced solutions (for automatic loading).
NONE	Suppresses writing of the specified results item for all substeps.
ALL	Writes the solution of the specified solution results item for every substep. This value is the default for a harmonic analysis (**ANTYPE**,HARMIC) and for any expansion pass (**EXPASS**,ON).
LAST	Writes the specified solution results item only for the last substep of each load step. This value is the default for a static (**ANTYPE**,STATIC) or transient (**ANTYPE**,TRANS) analysis.

%*array*%	Where *array* is the name of an *n* x 1 x 1 dimensional array parameter defining *n* key times, the data for the specified solution results item is written at those key times. Key times in the array parameter must appear in ascending order. Values must be greater than or equal to the beginning values of the load step, and less than or equal to the ending time values of the load step. For multiple-load-step problems, either change the parameter values to fall between the beginning and ending time values of the load step or erase the current settings and reissue the command with a new array parameter. For more information about defining array parameters, see the ***DIM** command documentation.

Cname
The name of the component, created with the **CM** command, defining the selected set of elements or nodes for which this specification is active. If blank, the set is all entities. A component name is not allowed with the ALL, BASIC, or RSOL items.

Command Default

The **OUTRES** command writes the specified solution results item for every substep. The exceptions to the default behavior are as follows:

- For static (**ANTYPE**,STATIC) or transient (**ANTYPE**,TRANS) analyses, the default is to write the specified solution results item for the last substep of each load step.
- For a harmonic analysis (**ANTYPE**,HARMIC) and any expansion pass, the default is to write the specified solution results item for every substep.
- For reduced or mode superposition transients, the default is to write the reduced displacements file for every 4th substep (as well as the last substep); however, if gap conditions exist, the default is to write the specified solution results item of every substep.

Notes

The **OUTRES** command allows you to specify the following:

- The solution item (*Item*) to write to the database (and to the reduced displacement and results files)
- The frequency (*FREQ*) at which the solution item is written (applicable to static, transient, or full harmonic analyses)
- The set of elements or nodes (*Cname*) to which your specification applies.

The command generates a specification for controlling data storage for each substep, activating storage of the specified results item for the specified substeps of the solution *and* suppressing storage of that item for *all other substeps*.

You can issue multiple **OUTRES** commands in an analysis. After the initial command creating the storage specification, subsequent **OUTRES** commands *modify* the specification set for each substep. The command processes your specifications at each substep in the order in which you input them. If you specify a given solution item twice, output is based upon the *last* specification. Therefore, issue multiple **OUTRES** commands carefully and in the proper sequence.

Besides **OUTRES**, another output-control command named **OUTPR** exists which controls solution *printout*. You can issue up to 50 output-control commands (either **OUTRES** or **OUTPR**, or some combination of both) in an analysis.

Note — Issuing **OUTRES**,ERASE erases the existing output specifications and resets the counted number of **OUTRES** commands to zero. Issuing **OUTPR**,ERASE affects the **OUTPR** command in the same way.

A given **OUTRES** command has no effect on results items *not* specified. For example, an **OUTRES**,ESOL,LAST command does not affect NSOL data; that is, it neither activates nor suppresses NSOL data storage in any substep.

Caution: In the results-item hierarchy, certain items are subsets of other items. For example, element solution (ESOL) data is a subset of all (ALL) solution data. Therefore, an **OUTRES**,ALL command can affect ESOL data. Likewise, an **OUTRES** command that controls ESOL data can affect a *portion* of ALL data. The example **OUTRES** commands illustrate the interrelationships between results items and the necessity for employing the **OUTRES** command thoughtfully.

To suppress ALL data at every substep, issue an **OUTRES**,ALL,NONE command. (An **OUTRES**,ERASE command *does not* suppress ALL data at every substep.)

The NSOL, RSOL, V, and A solution items are associated with nodes. All remaining solution items are associated with elements.

The boundary conditions (constraints and force loads) are written to the results file only if either nodal or reaction loads (NLOAD or RSOL items) are also written.

When specifying a *FREQ* value, observe the following:

- For a modal analysis, the only valid values are NONE or ALL.
- If you issue multiple **OUTRES** commands during an analysis, you cannot specify a key time array parameter (%*array*%) in a given **OUTRES** command and then specify a different *FREQ* option in a subsequent **OUTRES** command.

The **OUTRES** command is also valid in **/PREP7**.

Example

When issuing an **OUTRES** command, think of a matrix in which you set switches on and off. When a switch is on, a results item is stored for the specified substep. When a switch is off, a results item is suppressed for a specified substep.

Assuming a static (**ANTYPE**,STATIC) analysis, this example shows how the matrix looks after issuing each **OUTRES** command in this six-substep solution.

```
NSUBST,6
OUTRES,ERASE
OUTRES,NSOL,2
OUTRES,ALL,3
OUTRES,ESOL,4
SOLVE
```

To simplify the example, only a subset of the available solution items appears in the matrix.

OUTRES,ERASE -- After issuing this command, the default output specifications are in effect, as shown:

Substep	**Results Item Specification**						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
			NLOAD	STRS	FGRAD	EPEL	EPTH
1	off	off	off	off	off	off	off
2	off	off	off	off	off	off	off
3	off	off	off	off	off	off	off
4	off	off	off	off	off	off	off
5	off	off	off	off	off	off	off
6	**ON**	**ON**	**ON**	**ON**	**ON**	**ON**	**ON**

OUTRES,NSOL,2 -- This command modifies the initial specifications so that NSOL is turned on for substeps 2, 4 and 6, and turned off for substeps 1, 3 and 5, as shown:

Substep	**Results Item Specification**						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
			NLOAD	STRS	FGRAD	EPEL	EPTH
1	off	off	off	off	off	off	off
2	**ON**	off	off	off	off	off	off
3	off	off	off	off	off	off	off
4	**ON**	off	off	off	off	off	off
5	off	off	off	off	off	off	off
6	ON	ON	ON	ON	ON	ON	ON

OUTRES,ALL,3 -- This command further modifies the specifications so that ALL is turned on for substeps 3 and 6, and turned off for substeps 1, 2, 4 and 5, as shown:

Substep	**Results Item Specification**						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
			NLOAD	STRS	FGRAD	EPEL	EPTH
1	off	off	off	off	off	off	off
2	**off**	off	off	off	off	off	off
3	**ON**	**ON**	**ON**	**ON**	**ON**	**ON**	**ON**
4	**off**	off	off	off	off	off	off
5	off	off	off	off	off	off	off
6	ON	ON	ON	ON	ON	ON	ON

OUTRES,ESOL,4 -- This command once again modifies the specifications so that ESOL is turned on for the fourth and last substeps, and turned off for substeps 1, 2, 3 and 5, as shown:

Substep	**Results Item Specification**						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
			NLOAD	STRS	FGRAD	EPEL	EPTH
1	off	off	off	off	off	off	off
2	off	off	off	off	off	off	off
3	ON	ON	**off**	**off**	**off**	**off**	**off**
4	off	off	**ON**	**ON**	**ON**	**ON**	**ON**
5	off	off	off	off	off	off	off
6	ON	ON	ON	ON	ON	ON	ON

SOLVE

When obtaining the solution, results data is stored as follows:

Substep	Results Items Stored
1	*No data*
2	*No data*
3	NSOL and RSOL data
4	ESOL data
5	*No data*
6	ALL data

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic
Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>DB/Results File
Main Menu>Solution>Analysis Type>Sol'n Controls>Basic
Main Menu>Solution>Load Step Opts>Output Ctrls>DB/Results File

P Commands

PADELE, *DELOPT*

Deletes a defined path.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

DELOPT

Path delete option (one of the following):

ALL

Delete all defined paths.

NAME

Delete a specific path from the list of path definitions. (Substitute the actual path name for NAME.)

Command Default

Deletes the currently active path.

Notes

Paths are identified by individual path names. To review the current list of path names, issue the command **PATH**,STATUS.

This command is valid in the general postprocessor.

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Path from array
Main Menu>General Postproc>Path Operations>Delete Path>All Paths
Main Menu>General Postproc>Path Operations>Delete Path>By Name
Main Menu>Preprocessor>Path Operations>Delete Path>All Paths
Main Menu>Preprocessor>Path Operations>Delete Path>By Name

/PAGE, *ILINE*, *ICHAR*, *BLINE*, *BCHAR*

Defines the printout and screen page size.

POST1: Listing

MP ME ST DY <> PR EM <> FL PP ED

ILINE

Number of lines (11 minimum) per "page" or screen. Defaults to 24. Applies to interactive non-GUI to the screen output only.

ICHAR

Number of characters (41 to 132) per line before wraparound. Defaults to 80. Applies to interactive non-GUI to the screen output only.

BLINE
Number of lines (11 minimum) per page. Defaults to 56. Applies to batch mode [**/BATCH**], diverted [**/OUTPUT**], or interactive GUI [**/MENU**] output.

BCHAR
Number of characters (41 to 240 (system dependent)) per line before wraparound. Defaults to 132. Applies to batch mode [**/BATCH**], diverted [**/OUTPUT**], or interactive GUI [**/MENU**] output.

Command Default

As defined by the items above.

Notes

Defines the printout page size for batch runs and the screen page size for interactive runs. Applies to the POST1 **PRNSOL**, **PRESOL**, **PRETAB**, **PRRSOL**, and **PRPATH** commands. See the **/HEADER** command for additional controls (page ejects, headers, etc.) that affect the amount of printout. A blank (or out-of-range) value retains the previous setting. Issue **/PAGE**,STAT to display the current settings. Issue **/PAGE**,DEFA to reset the default specifications.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

PAGET, *PARRAY*, *POPT*

Writes current path information into an array variable.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

PARRAY
The name of the array parameter that the ANSYS program creates to store the path information. If the array parameter already exists, it will be replaced with the current path information.

POPT
Determines how data will be stored in the parameter specified with *PARRAY*:

POINTS
Store the path points, the nodes (if any), and coordinate system. (For information on defining paths and path points, see the descriptions of the **PATH** and **PPATH** commands.)

TABLE
Store the path data items. (See the **PDEF** command description for path data items.)

LABEL
Stores path data labels.

Notes

Use the **PAGET** command together with the **PAPUT** command to store and retrieve path data in array variables for archiving purposes. When retrieving path information, restore the path points (POINTS option) first, then the path data (TABLE option), and then the path labels (LABEL option).

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Store>Path in array

---

PAPUT, *PARRAY*, *POPT*

Retrieves path information from an array variable.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

PARRAY

Name of the array variable containing the path information.

P0PT

Specifies which path data to retrieve:

POINTS

Retrieve path point information (specified with the **PPATH** command and stored with the **PAGET**,POINTS command). The path data name will be assigned to the path points.

TABLE

Retrieve path data items (defined via the **PDEF** command and stored with the **PAGET**,,TABLE command).

LABEL

Retrieve path labels stored with the **PAGET**,,LABEL command.

Notes

When retrieving path information, restore path points (POINTS option) first, then the path data (TABLE option), and then the path labels (LABEL option).

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Path from array

---

PARESU, *Lab*, *Fname*, *Ext*, --

Restores previously saved paths from a file.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

Lab

Read operation:

ALL
Read all paths from the selected file (default).

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to PATH if *Fname* is blank.

--
Unused field

Notes

This command removes all paths from virtual memory and then reads path data from a file written with the **PASAVE** command. All paths on the file will be restored. All paths currently in memory will be deleted.

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Paths from file
Main Menu>Preprocessor>Path Operations>Retrieve>Paths from file

---

PARTSEL, *Type*, *PMIN*, *PMAX*, *PINC*

Selects a subset of parts in an explicit dynamic analysis.

DATABASE: Selecting

<> <> <> DY <> <> <> <> <> <> <>

Type
Label identifying type of select. Because **PARTSEL** is a command macro, the label must be enclosed in single quotes.

'S'
Select a new set (default).

'R'
Reselect a set from the current set.

'A'
Additionally select a set and extend the current set.

'U'
Unselect a set from the current set.

'ALL'
Select all parts.

'NONE'
Unselect all parts.

'INVE'
Invert the current selected set.

The following fields are used only with *Type* = 'S', 'R', 'A', or 'U':

PMIN
Minimum part number in the range to be selected.

PMAX
Maximum part number in the range to be selected (defaults to *PMIN*).

PINC
Part number increment in the range to be selected (default =1).

Notes

PARTSEL invokes an ANSYS macro that selects parts in an explicit dynamic analysis. When **PARTSEL** is executed, an element component is automatically created for each existing part. For example, the elements that make up PART 1 are grouped into the element component _PART1. Each time the **PARTSEL** command is executed, components for unselected parts will be unselected. To plot selected parts, choose **Utility Menu> Plot> Parts** in the GUI or issue the command **PARTSEL**,'PLOT'.

After selecting parts, if you change the selected set of nodes or elements and then plot parts, the nodes and elements associated with the previously selected parts (from the last **PARTSEL** command) will become the currently selected set.

Note — A more efficient way to select and plot parts is to use the **ESEL** (with *ITEM* = PART) and **EPLOT** commands. We recommend using **ESEL** instead of **PARTSEL** since **PARTSEL** will be phased out in a future release. Note that the menu path mentioned above for plotting parts does not work with the **ESEL** command; use **Utility Menu> Plot> Elements** instead.

In an explicit dynamic small restart analysis (**EDSTART**,2), **PARTSEL** can be used to unselect a part during the solution even if it is referenced in some way (such as in a contact definition). (Note that **ESEL** cannot be used for this purpose.) However, in a new analysis or a full restart analysis (**EDSTART**,3), all parts that are used in some type of definition must be selected at the time of solution.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Select Parts

PASAVE, *Lab*, *Fname*, *Ext*, --

Saves selected paths to an external file.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

Lab
Write operation:

S
Saves only selected paths.

ALL
Saves all paths (default).

Pname
Saves the named path (from the **PSEL** command).

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to PATH if *Fname* is blank.

--
Unused field

Notes

Saves the paths selected with the **PSEL** command to an external file (**Jobname.path** by default). Previous paths on this file, if any, will be overwritten. The path file may be read with the **PARESU** command.

This command is valid in /Post1.

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Store>Paths in file
Main Menu>Preprocessor>Path Operations>Store>Paths in file

PATH, *NAME*, *nPts*, *nSets*, *nDiv*

Defines a path name and establishes parameters for the path.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

NAME
Name for this path (eight characters maximum. If *nPts* is blank, set the current path to the path with this name. If *nPts* is greater than zero, create a path of this name. If a path with this name already exists, replace it with a new path. If the *NAME* value is STATUS, display the status for path settings.

nPts
The number of points used to define this path. The minimum number is two, and the maximum is 1000.

nSets
The number of sets of data which you can map to this path. You must specify at least four: X, Y, Z, and S. Default is 30.

nDiv
The number of divisions between adjacent points. Default is 20. There is no maximum number of divisions.

Notes

The **PATH** command is used to define parameters for establishing a path. The path geometry is created by the **PPATH** command. Multiple paths may be defined and named; however, only one path may be active for data interpolation [**PDEF**] and data operations [**PCALC**, etc.]. Path geometry points and data are stored in memory while in POST1. If you leave POST1, the path information is erased. Path geometry and data may be saved in a file by archiving the data using the **PASAVE** command. Path information may be restored by retrieving the data using the **PARESU** command.

For overlapping nodes, the lowest numbered node is assigned to the path.

The number of divisions defined using *nDiv* does NOT affect the number of divisions used by **PLSECT** and **PRSECT**.

For information on displaying paths you have defined, see the *ANSYS Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>List Results>Path Items
Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Path from array
Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Paths from file
Main Menu>General Postproc>Path Operations>Define Path>By Location
Main Menu>General Postproc>Path Operations>Define Path>By Nodes
Main Menu>General Postproc>Path Operations>Define Path>On Working Plane
Main Menu>General Postproc>Path Operations>Define Path>Path Status>Defined Paths
Main Menu>General Postproc>Path Operations>Delete Path>All Paths
Main Menu>General Postproc>Path Operations>Delete Path>By Name
Main Menu>General Postproc>Path Operations>Recall Path
Main Menu>Preprocessor>Path Operations>Define Path>By Location
Main Menu>Preprocessor>Path Operations>Define Path>By Nodes
Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane
Main Menu>Preprocessor>Path Operations>Define Path>Path Status>Defined Paths
Main Menu>Preprocessor>Path Operations>Delete Path>All Paths
Main Menu>Preprocessor>Path Operations>Delete Path>By Name
Main Menu>Preprocessor>Path Operations>Recall Path
Main Menu>Preprocessor>Path Operations>Retrieve>Paths from file
Utility Menu>List>Status>General Postproc>Path Operations

/PBC, *Item*, --, *KEY*, *MIN*, *MAX*, *ABS*

Shows boundary condition (BC) symbols and values on displays.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

Item
Label identifying the item:

U
Applied translational constraints (UX, UY, UZ).

ROT
Applied rotational constraints (ROTX, ROTY, ROTZ).

TEMP
Applied temperatures (TEMP, TBOT, TE2, TE3, . . ., TTOP).

PRES
Applied fluid pressures.

V
Applied flow velocities (VX, VY, VZ).

SP0*n*
Mass fraction of species *n*, where *n* = 1 to 6 (FLOTRAN). If a species is given a user-defined name [**MSSPEC**], use that name instead of SP0*n*.

ENKE
Turbulent kinetic energy (FLOTRAN).

ENDS
Turbulent energy dissipation (FLOTRAN).

VOLT
Applied voltages.

MAG
Applied scalar magnetic potentials.

A
Applied vector magnetic potentials.

CHRG
Applied electric charge.

F or FORC
Applied structural forces (FX, FY, FZ).

M or MOME
Applied structural moments (MX, MY, MZ).

HEAT
Applied heat flows (HEAT, HBOT, HE2, HE3, . . ., HTOP).

FLOW
Applied fluid flow.

AMPS
Applied current flow.

FLUX
Applied magnetic flux.

CSG
Applied magnetic current segments.

MAST
Master degrees of freedom.

CP
Coupled nodes.

CE
Nodes in constraint equations.

NFOR
POST1 nodal forces.

NMOM
POST1 nodal moments

RFOR
POST1 reaction forces.

RMOM
POST1 reaction moments (MX, MY, MZ).

PATH
Path geometry (undistorted) associated with the **PATH** command after a **PDEF** or **PVECT** command has been issued.

ACEL
Global acceleration (ACELX, ACELY, ACELZ vector).

OMEG
Global angular velocity (OMEGX, OMEGY, OMEGZ vector).

WELD
Applied spotwelds (ANSYS LS-DYNA).

ALL
Represents all appropriate labels.

`--`
Unused field.

KEY
Symbol key:

0
Do not show symbol.

1
Show symbol.

2
Plot value next to symbol.

MIN
Minimum value in a range of values plotted on screen.

MAX
Maximum value in a range of values plotted on screen.

ABS
Absolute number. If *KEY* = 2 and *ABS* = 0, a number falling between the *MIN* and *MAX* is displayed. If *ABS* is not specified, it defaults to 0. If *KEY* = 2 and *ABS* = 1, an absolute value falling between the *MIN* and *MAX* is displayed. *ABS* = 1 lets you eliminate the display of numbers whose absolute values are less than a desired tolerance. For example, if *ABS* = 1, *MIN* = 10 and *MAX* = 1e8, values such as .83646 and -5.59737 are not displayed.

Command Default

No symbols displayed.

Notes

The **/PBC** command adds degree of freedom constraint, force load, and other symbols to displays.

Symbols are applied to the selected nodes only. All arrow and arrowhead symbols are oriented in the nodal coordinate system and lie in two perpendicular planes. Force arrows are scaled proportional to their magnitude. (If *KEY* = 1, use **/VSCALE** to change arrow length.) For scalar quantities, the specific component direction (i.e., x, y, or z) of the symbol has no meaning, but the positive or negative sense (e.g., positive or negative x) represents a positive or negative scalar value, respectively.

The effects of the **/PBC** command are not cumulative (that is, the command does not modify an existing setting from a previously issued **/PBC** command). If you issue multiple **/PBC** commands during an analysis, only the setting specified by the most recent **/PBC** command applies.

Use **/PSTATUS** or **/PBC**,STAT to display settings. Use **/PBC**,DEFA to reset all specifications back to default. See the **/PSF** and **/PBF** commands for other display symbols.

In a cyclic symmetry analysis, the **/PBC** command is deactivated when cyclic expansion is active (**/CYCEXPAND**,,ON). To view boundary conditions on the basic sector, deactivate cyclic expansion (**/CYCEXPAND**,,OFF) and issue this command: **/PBC**,ALL,,1

Issuing the command **/PBC**,PATH, ,1 displays all defined paths.

The **/PBC** command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane
Main Menu>General Postproc>Path Operations>Plot Paths
Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane
Main Menu>Preprocessor>Path Operations>Plot Paths
Utility Menu>PlotCtrls>Symbols

/PBF, *Item*, --, *KEY*

Shows magnitude of body force loads on displays.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

Item
Label identifying the item:

TEMP
Applied temperatures.

FLUE
Applied fluences.

HGEN
Applied heat generation rates.

JS
Applied current density magnitude.

JSX
X-component of current density.

JSY
Y-component of current density.

JSZ
Z-component of current density.

PHASE
Phase angle of applied load.

MVDI
Applied magnetic virtual displacements flag.

CHRGD
Applied electric charge density.

VLTG
Applied voltage drop.

FORC
Applied force density (FLOTRAN only).

- -
Unused field.

KEY
Symbol key:

0
Do not show body force load contours.

1
Show body force load contours.

2
Show current density as a vector (not a contour).

Command Default

No body force load contours displayed.

Notes

Shows body force loads as contours on displays for the selected elements.

The effects of the **/PBF** command are not cumulative (that is, the command does not modify an existing setting from a previously issued **/PBF** command). If you issue multiple **/PBF** commands during an analysis, only the setting specified by the most recent **/PBF** command applies.

Use **/PSTATUS** or **/PBF**,STAT to display settings. Use **/PBF**,DEFA to reset all specifications back to default. See also the **/PSF** and **/PBC** command for other display contours.

Portions of this command are not supported by PowerGraphics [**/GRAPHICS**,POWER].

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Symbols

PCALC, *Oper*, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*, *CONST*

Forms additional labeled path items by operating on existing path items.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

Oper
Type of operation to be performed. See Notes below for specific descriptions of each operation:

ADD
Adds two existing path items.

MULT
Multiplies two existing path items.

DIV
Divides two existing path items (a divide by zero results in a value of zero).

EXP
Exponentiates and adds existing path items.

DERI
Finds a derivative.

INTG
Finds an integral.

SIN
Sine.

COS
Cosine.

ASIN
Arcsine.

ACOS
Arccosine.

LOG
Natural log.

LabR
Label assigned to the resulting path item.

Lab1
First labeled path item in operation.

Lab2
Second labeled path item in operation. *Lab2* must not be blank for the MULT, DIV, DERI, and INTG operations.

FACT1
Factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

FACT2
Factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

CONST
Constant value (defaults to 0.0).

Notes

If *Oper* = ADD, the command format is:

PCALC,ADD,*LabR*,*Lab1*,*Lab2*,*FACT1*,*FACT2*,*CONST*

This operation adds two existing path items according to the operation:

$$LabR = (FACT1 \times Lab1) + (FACT2 \times Lab2) + CONST$$

It may be used to scale the results for a single path item.

If *Oper* = MULT, the command format is:

PCALC,MULT,*LabR*,*Lab1*,*Lab2*,*FACT1*

Lab2 must not be blank. This operation multiplies two existing path items according to the operation:

$$LabR = Lab1 \times Lab2 \times FACT1$$

If *Oper* = DIV, the command format is:

PCALC,DIV,*LabR*,*Lab1*,*Lab2*,*FACT1*

Lab2 must not be blank. This operation divides two existing path items according to the operation:

$$LabR = (Lab1/Lab2) \times FACT1$$

If *Oper* = EXP, the command format is:

PCALC,EXP,*LabR*,*Lab1*,*Lab2*,*FACT1*,*FACT2*

This operation exponentiates and adds existing path items according to the operation:

$$LabR = (|Lab1|^{FACT1}) + (|Lab2|^{FACT2}|)$$

If *Oper* = DERI, the command format is:

PCALC,DERI,*LabR*,*Lab1*,*Lab2*,*FACT1*

Lab2 must not be blank. This operation finds a derivative according to the operation:

$$LabR = FACT1 \times d(Lab1)/d(Lab2)$$

If *Oper* = INTG, the command format is:

PCALC,INTG,*LabR*,*Lab1*,*Lab2*,*FACT1*

Lab2 must not be blank. This operation finds an integral according to the operation:

$$LabR = FACT1 \times \int_{S} Lab1 d(Lab2)$$

Use S for *Lab2* to integrate *Lab1* with respect to the path length. S, the distance along the path, is automatically calculated by the program when a path item is created with the **PDEF** command.

If *Oper* = SIN, COS, ASIN, ACOS, or LOG, the command format is:

PCALC,Oper,*LabR*,*Lab1*,,*FACT1*,*CONST*

where the function (SIN, COS, ASIN, ACOS or LOG) is substituted for *Oper* and *Lab2* is blank.

The operation finds the resulting path item according to one of the following formulas:

LabR = FACT2 x *sin(FACT1 x Lab1) + CONST*

LabR = FACT2 x *cos(FACT1 x Lab1) + CONST*

LabR = FACT2 x sin^{-1}*(FACT1 x Lab1) + CONST*

LabR = FACT2 x cos^{-1}*(FACT1 x Lab1) + CONST*

LabR = FACT2 x *log(FACT1 x Lab1) + CONST*

Menu Paths

Main Menu>General Postproc>Path Operations>Add
Main Menu>General Postproc>Path Operations>ArcCosine
Main Menu>General Postproc>Path Operations>ArcSine
Main Menu>General Postproc>Path Operations>Cosine
Main Menu>General Postproc>Path Operations>Differentiate
Main Menu>General Postproc>Path Operations>Divide
Main Menu>General Postproc>Path Operations>Exponentiate
Main Menu>General Postproc>Path Operations>Integrate
Main Menu>General Postproc>Path Operations>Multiply
Main Menu>General Postproc>Path Operations>Natural Log
Main Menu>General Postproc>Path Operations>Sine

PCIRC, *RAD1*, *RAD2*, *THETA1*, *THETA2*

Creates a circular area centered about the working plane origin.

PREP7: Primitives

MP ME ST DY <> PR EM <> FL PP ED

RAD1, *RAD2*

Inner and outer radii (either order) of the circle. A value of either zero or blank for either *RAD1* or *RAD2*, or the same value for both *RAD1* and *RAD2*, defines a solid circle.

THETA1, *THETA2*

Starting and ending angles (either order) of the circular area. Used for creating a circular sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle.

The starting angle defaults to 0.0° and the ending angle defaults to 360.0°. See the *ANSYS Modeling and Meshing Guide* for an illustration.

Notes

Defines a solid circular area or circular sector centered about the working plane origin. For a solid circle of 360°, the area will be defined with four keypoints and four lines. See the **CYL4** and **CYL5** commands for alternate ways to create circles.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Circle>By Dimensions

---

/PCIRCLE, *XCENTR*, *YCENTR*, *XLRAD*

Creates an annotation circle (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

XCENTR
Circle X center location (-1.0 < X < 2.0).

YCENTR
Circle Y center location (-1.0 < Y < 1.0).

XLRAD
Circle radius length.

Notes

Creates an annotation circle to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All circles are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** and the **/PSPEC** command to set the attributes of the circle.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

PCONV, *TOLER*, *Item*, *Comp*, *NODE*, *Surf*

Sets convergence values for p-method solutions.

SOLUTION: p-Method

MP ME ST <> <> PR <> <> <> PP ED

TOLER

Tolerance (as a percentage) used for convergence checking. The analysis is considered to be converged if the values of all specified items change by less than *TOLER* at the end of a p-loop. Defaults to 5. If *TOLER* = STAT, list the currently specified p-convergence criteria. If *TOLER* = ERASE, erase all p-convergence specifications. If *TOLER* = DELE, delete only the tolerance for the specification denoted by the *Item*, *Comp*, *NODE*.

Item

Convergence based on (Valid component labels are described in PCONV - Valid Item and Component Labels below):

SE
: Strain energy or stored electrostatic energy (default). Does not require a component label.

S
: Stress.

EPEL
: Strain.

U
: Translational displacement.

ROT
: Structural rotation. (Valid only for SHELL150.)

VOLT
: Electric potential.

EF
: Electric field strength.

D
: Electric flux density.

EFORC
: Global electrostatic (Maxwell Stress Tensor) force.

Comp

Component of the *Item*. (Not required for *Item* = SE, VOLT, or EFORC). If *Item* = S or EPEL, *Comp* defaults to EQV; if *Item* = U, ROT, EF, or D, *Comp* defaults to SUM. Valid component labels for each *Item* are described in PCONV - Valid Item and Component Labels below.

NODE

Node number where convergence checking is to be performed. Valid only for *Item* = S, EPEL, U, ROT, VOLT, EF, or D. For valid *Item* labels, a node number must be entered in this field. If *NODE* = P, graphical picking is enabled (valid only in the GUI). A component name may be substituted for *NODE*.

Surf

Surface of a p-element shell structure where the convergence checking for *NODE* will be performed. Valid only for *Item* = S or EPEL.

TOP
Top surface (default).

MID
Middle surface.

BOT
Bottom surface.

Command Default

Convergence is checked between successive p-loops by comparing the change in strain energy (SE) to *TOLER*.

Notes

When you use the menu paths to delete p-method convergence criteria, the ANSYS program writes the command **PCONV**,DELE,*n* to its log file.

The p-level (polynomial level, i.e., the order or level *n* of a polynomial, where $2 \leq n \leq 8$) is increased for each loop until the values of the specified items (*Item*,*Comp*) change less than the specified tolerance (*TOLER*) between loops, or until the maximum p-level has been reached [**PPRANGE**]. Convergence checking is performed only on included elements [**PINCLUDE**]. Up to 10 separate specifications may be activated at one time. Valid *Item* and *Comp* labels are shown next.

PCONV - Valid Item and Component Labels

Item	Comp	Description
SE		Strain energy or stored electrostatic energy
S	X, Y, Z, XY, YZ, XZ	Component stress
"	1, 2, 3	Principal stress
"	INT	Stress intensity
"	EQV	Equivalent stress
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain
"	1, 2, 3	Principal elastic strain
"	INT	Elastic strain intensity
"	EQV	Elastic equivalent strain
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum (valid only for SHELL150)
VOLT		Electric potential
EF	X, Y, Z, SUM	Electric field strength
D	X, Y, Z, SUM	Electric flux density
EFORC	X, Y, Z, SUM	Global electrostatic (Maxwell Stress Tensor) force

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>p-Method>Convergence Crit
Main Menu>Solution>Load Step Opts>p-Method>Convergence Crit

/PCOPY, *KEY*

Automatically generates hard copies for HP UNIX work stations.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

KEY

Copy key:

0

No specification setting for automatic hard copy of display.

1

Set specification for automatic hard copy after each display.

NOW

(Action) Produce hard copy of current display (*KEY* is not reset to 1).

Command Default

No automatic hard copy of display.

Notes

Sets automatic hard copy specification. This command is available only on HP work stations, and only during interactive runs with the **/SHOW** specification active (for terminals with hard copy capability).

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

PCORRO, *CTK*

Specifies the allowable exterior corrosion thickness for a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

CTK

Allowable corrosion thickness.

Notes

Specifies the allowable exterior corrosion thickness for a piping run. See the PREP7 **RUN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Specifications

---

PCROSS, *LabXR*, *LabYR*, *LabZR*, *LabX1*, *LabY1*, *LabZ1*, *LabX2*, *LabY2*, *LabZ2*

Calculates the cross product of two path vectors along the current path.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

LabXR
Label assigned to X-component of resultant vector.

LabYR
Label assigned to Y-component of resultant vector.

LabZR
Label assigned to Z-component of resultant vector.

LabX1
X-component of first vector label (labeled path item).

LabY1
Y-component of first vector label.

LabZ1
Z-component of first vector label.

LabX2
X-component of second vector label (labeled path item).

LabY2
Y-component of second vector label.

LabZ2
Z-component of second vector label.

Menu Paths

Main Menu>General Postproc>Path Operations>Cross Product

PDANL, *Fname*, *Ext*, --

Defines the analysis file to be used for probabilistic looping.

PROBABILISTIC: Deterministic Model

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Ext
Filename extension (8 character maximum).

--
Unused field

Command Default

If interactive, no default; if batch, use the batch input stream (**File.BAT**).

Notes

The analysis file containing the deterministic, parameterized model must be specified if the probabilistic analysis is performed interactively. The file must exist at the time the **PDANL** command is issued. In this file, where **/PREP7** and **/PDS** occur, they must be the first nonblank characters on the line (do not use the $ delimiter on any **/PREP7** and **/PDS** command lines).

By default the analysis files specified with **PDANL** are executed from the top. All definitions of random input variables (for example, APDL parameters defined as random input variables with the **PDVAR** command, using ***SET** or *Pname* = ...) are ignored in the analysis file. The PDS takes control of setting the values of random input variable values for each loop.

Menu Paths

Main Menu>Prob Design>Analysis File>Assign

PDCDF, *Rlab*, *Name*, *Type*, *CONF*, *NMAX*

Plots the cumulative distribution function.

PROBABILISTIC: Postprocessing

MP ME ST DY <> PR EM <> FL PP ED

Rlab

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an **RSFIT** command (for Response Surface Analyses).

Name

Parameter name. The parameter must have been previously defined as a random input variable or a random output parameter with the **PDVAR** command.

Type

Type of cumulative distribution curve to be displayed.

EMP

Show an empirical cumulative distribution curve.

GAUS

Show a cumulative distribution curve in a normal distribution plot. A random variable based on a normal or Gaussian distribution appears as a straight line in this type of plot.

LOGN

Show a cumulative distribution curve in a log-normal plot. A random variable based on a log-normal distribution appears as a straight line in this type of plot.

WEIB

Show a cumulative distribution curve in a Weibull distribution plot. A random variable based on a Weibull distribution appears as a straight line in this type of plot.

CONF

Confidence level. The confidence level is used to plot confidence bounds around the cumulative distribution function. The value for the confidence level must be between 0.0 and 1.0 and it defaults to 0.95 (95%). Plotting

of the confidence bound is suppressed for *CONF* ≤ 0.5. This parameter is ignored for the postprocessing of response surface methods results.

NMAX
Maximum number of points to be plotted for the distribution curve. This must be a positive number and it defaults to 100. If the sample size is less than *NMAX*, all sample data is represented in the plot. If the sample size is larger than *NMAX*, the probabilistic design system classifies the sample into *NMAX* classes of appropriate size.

Command Default

Rlab as described above, *Type* = EMP, *CONF* = 0.95, *NMAX* = 100.

Notes

Plots the cumulative distribution function.

The **PDCDF** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

Menu Paths

Main Menu>Prob Design>Prob Results>Statistics>CumulativeDF

PDCFLD, *ParR*, *Entity*, *Ctype*, *CLENGTH*

Calculates a correlation field and stores it into an ANSYS array.

PROBABILISTIC: Preprocessing

MP ME ST DY <> PR EM <> FL PP ED

ParR
Parameter name. *ParR* is a one-dimensional array with the dimension *N* * (*N* - 1)/2, where *N* is either the number of the selected nodes or the number of the selected elements (depending on the *Entity* field). The **PDCFLD** command automatically sets *ParR* as a one-dimensional array, (so you do not have to use the ***DIM** command). If you use the **PDCFLD** command twice with the ANSYS parameter *ParR*, then the values stored in the array are automatically overwritten. If the number of selected FE entities is different from the previous **PDCFLD** command, then the array *ParR* is re-dimensioned automatically.

Entity
Specifies which FE entity the calculation of the correlation field is based on. This field must not be blank.

NODE
Calculate the correlation coefficients based on the distance between the selected nodes.

ELEM
Calculate the correlation coefficients based on the distance between the centroids of the selected elements.

Ctype

Specifies the equation used to calculate the correlation coefficients as a function of the nodal or element centroid distances. This field must not be blank.

NONE

The random field is not correlated. This means the correlation coefficients are determined according to

$\rho_{ij} = 1$ for $i = j$

$\rho_{ij} = 0$ for $i \neq j$

Here, ρ_{ij} is the correlation coefficient between the i-th and j-th selected FE entity (node or element centroid).

LEXP

Calculate the correlation coefficient according to a linear-exponential decay function.

$$\rho_{ij} = \exp\left(-\frac{D(\{x_i\},\{x_j\})}{C_L}\right)$$

Here, $D(\{x_i\}, \{x_j\})$ is the "domain distance" between $\{x_i\}$, $\{x_j\}$, and $\{x_i\}$ and $\{x_j\}$ are the coordinate vectors of the i-th and j-th selected FE entity (node or element centroid), and C_L is the correlation length of the random field as specified in the *CLENGTH* field.

QEXP

The correlation coefficient is calculated according to a quadratic- exponential decay function.

$$\rho_{ij} = \exp\left(-\left(\frac{D(\{x_i\},\{x_j\})}{C_L}\right)^2\right)$$

Here, $D(\{x_i\}, \{x_j\})$ is the "domain distance" between $\{x_i\}$, $\{x_j\}$, and $\{x_i\}$ and $\{x_j\}$ are the coordinate vectors of the i-th and j-th selected FE entity (node or element centroid), and C_L is the correlation length of the random field as specified in the *CLENGTH* field.

DIST

Calculate only $D(\{x_i\}, \{x_j\})$. $D(\{x_i\}, \{x_j\})$ is the "domain distance" between $\{x_i\}$, $\{x_j\}$, and $\{x_i\}$ and $\{x_j\}$ are the coordinate vectors of the i-th and j-th selected FE entity (node or element centroid). The *CLENGTH* argument is ignored for this option.

CLENGTH

Correlation length of the correlation field. The correlation length is a characteristic length that influences how strongly two elements of a random field are correlated with each other. The larger the value of *CLENGTH*, the stronger the correlation between the random field elements. *CLENGTH* is required for *Ctype* = LEXP and *Ctype* = QEXP; it must be a nonzero, positive number.

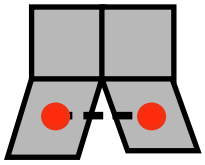
Notes

Calculates a correlation field for a probabilistic analysis involving a random field. Random fields are random effects with a spatial distribution; the value of a random field not only varies from simulation to simulation at any given location, but also from location to location. The correlation field describes the correlation coefficient between two different spatial locations. Random fields can be either based on element properties (typically material) or nodal properties (typically surface shape defined by nodal coordinates). Hence, random fields are either associated

with the selected nodes or the selected elements. If a random field is associated with elements, then the correlation coefficients of the random field are calculated based on the distance of the element centroids.

For more information, see Probabilistic Design in the *ANSYS Advanced Analysis Techniques Guide*.

Note that for correlation fields, the "domain distance" $D(\{x_i\}, \{x_j\})$ is not the spatial distance $|\{x_i\} - \{x_j\}|$, but the length of a path between $\{x_i\}$ and $\{x_j\}$ that always remains inside the finite element domain. However, exceptions are possible in extreme meshing cases. For elements that share at least one node, the **PDCFLD** evaluates the distance by directly connecting the element centroids with a straight line. If these neighboring elements form a sharp inward corner then it is possible that the "domain distance" path lies partly outside the finite element domain, as illustrated below.



After the correlation coefficients have been calculated and stored in the ANSYS parameter *ParR*, then use the **PDCORR** command to define the correlations between the elements of the random field.

Menu Paths

Main Menu>Prob Design>Prob Definitns>Correl Field

PDCLR, *Type*

Clears the probabilistic design database.

PROBABILISTIC: Database

MP ME ST DY <> PR EM <> FL PP ED

Type

Specifies the part of the probabilistic database to be cleared.

ALL

Clear the entire probabilistic database. Both the preprocessing and postprocessing parts are cleared.

POST

Clear only the postprocessing part of the probabilistic database. This is necessary if you want to modify the deterministic model (the analysis file) or probabilistic model (random variables, correlations between random variables, or the random output parameter) after a probabilistic analysis has already been performed.

Notes

Clears the probabilistic design system (PDS) database. The settings are reset to their default values and the memory is cleared. Remember that the result files containing the results of the probabilistic loops are never deleted unless you intentionally delete them. We recommend that you use this command before switching to a new probabilistic analysis using a different probabilistic model (a different analysis loop file or deterministic model), or changing random input variables, their correlations, or random output parameters. Clearing the probabilistic database is not necessary if the probabilistic model remains the same and will be analyzed with a different

probabilistic method or different method options. See the **PDEXE** command for restrictions. Before issuing the **PDCLR** command, you should save the probabilistic database using the **PDSAVE** command.

Menu Paths

Main Menu>Prob Design>Prob Database>Clear & Reset

PDCMAT, *Rlab*, *Matrix*, *Name1*, *Name2*, *Corr*, *SLEVEL*, *Popt*

Prints the correlation coefficient matrix.

PROBABILISTIC: Postprocessing

MP ME ST DY <> PR EM <> FL PP ED

Rlab

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an **RSFIT** command (for Response Surface Analyses).

Matrix

Keyword for the type of correlation coefficient matrix.

IO
: Matrix of correlation coefficients between random input variables and output parameters.

II
: Matrix of correlation coefficients between random input variables and other random input variables

OO
: Matrix of correlation coefficients between random output parameters and other random output parameters.

S
: Correlation coefficient between a single random parameter (input or output) and another random parameter (input or output). The probabilistic design parameters must be specified in *Name1* and *Name2* for this option.

Name1, *Name2*

Parameter names. The parameters must have been previously defined as a random input variable or a random output parameter with the **PDVAR** command. These parameters are used for *Matrix* = S only and are ignored for the other *Matrix* keywords.

Corr

Keyword for the type of correlation coefficients to be used for the output.

RANK
: Spearman rank-order correlation coefficient (default).

LIN
: Pearson linear correlation coefficient.

SLEVEL

Significance level. The value for the significance level must be between 0.0 and 1.0. The default value is 0.025 (2.5%).

Popt
Specifies whether the probabilities should be printed with the correlation coefficients.

0
Print only the correlation coefficients.

1
Print both the correlation coefficients and the probabilities (default).

Command Default

Rlab as described above, *Matrix* = IO, *Corr* = RANK, *SLEVEL* = 0.025, *Popt* = 1

Notes

Prints the correlation coefficient matrix.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

For all correlation coefficients the probabilistic design tool evaluates the probability that the correlation coefficient can be neglected. The evaluation of this probability is based on statistical test theory. The larger this probability is the likelier it is that the correlation coefficient does not really reflect an observable statistical interdependence between the parameters involved. If this probability exceeds the significance level as specified by the *SLEVEL* parameter, the correlation coefficient should be regarded as negligible or insignificant. The higher the significance level *SLEVEL*, the more correlation coefficients are considered significant. Using the *Popt* parameter you can also get a list of the probabilities and review them as to how far they exceed the significance level or how far they stay below it.

The **PDCMAT** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

Menu Paths

Main Menu>Prob Design>Prob Results>Trends>Correl Matrix

PDCORR, *Name1*, *Name2*, *CORR*

Specifies the correlation between two random input variables.

PROBABILISTIC: Preprocessing

MP ME ST DY <> PR EM <> FL PP ED

Name1
Parameter name. The parameter must have been previously defined as a random input variable with the **PDVAR** command.

Name2
Parameter name. The parameter must have been previously defined as a random input variable with the **PDVAR** command. Must be different from *Name1*.

CORR
Specification of the correlation:

Value
: Sets the correlation coefficient between *Name1* and *Name2* to the specified value. If this correlation coefficient was already defined it will be changed to this new value. The correlation coefficient must be between -1.0 and +1.0.

DEL
: Delete the previously defined correlation between *Name1* and *Name2*.

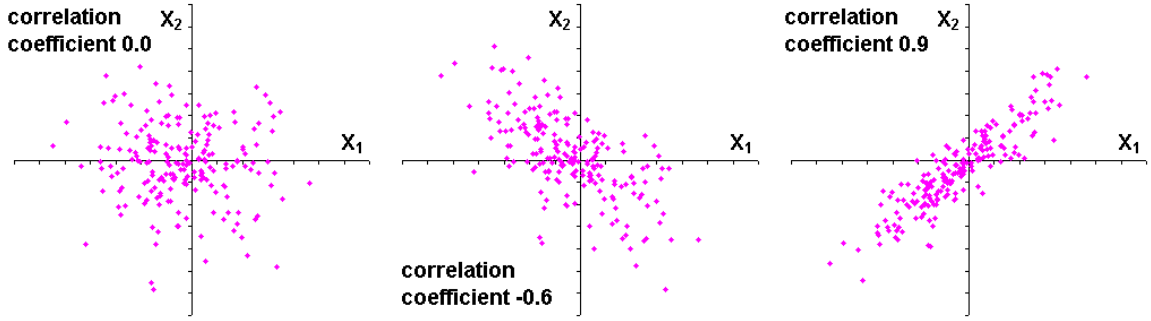
Command Default

No correlations defined between random input variables.

Notes

Specifies the correlations between two random input variables. The PDS tool generates correlated samples if correlations exist. This applies to both probabilistic methods (Monte Carlo Simulation and Response Surface Methods).

If there are correlated random input variables, the sampled locations of the random input variables reflect the correlation as illustrated below for the case of two standard normal distributed variables X_1 and X_2. The illustrations show no correlation (correlation coefficient 0.0), a relatively moderate negative correlation (correlation coefficient -0.6), and a relative strong positive correlation (correlation coefficient 0.9).



Menu Paths

Main Menu>Prob Design>Prob Definitns>Correlation

PDDMCS, *NSIM*, --, *Astop*, *ACCMEAN*, *ACCSTDEV*, *CHECK*, *Seed*

Specifies options for Monte Carlo Simulations using direct sampling.

PROBABILISTIC: Methods

MP ME ST DY <> PR EM <> FL PP ED

NSIM
: Number of simulation loops of the analysis.

--
: Unused field.

Astop
: Autostop option label.

AUTO
Enable Autostop. When Autostop is used, the PDS feature continues the simulation loops until the convergence criteria for the mean value and the standard deviation have been met or until the number of simulations *NSIM* are complete, whichever comes first. The convergence criteria (mean value and standard deviations of all random output parameters) are specified by the *ACCMEAN* and *ACCSTDEV* parameters. The criteria are met if the mean value *and* the standard deviations converge within the accuracy specified in the *ACCMEAN* and *ACCSTDEV* options. The convergence check is done every *i*-th loop, where *i* is specified in the *CHECK* parameter.

ALL
Disable Autostop option. All Monte Carlo Simulations as specified by *NSIM* are performed (default).

ACCMEAN
Accuracy of the mean values of all random output parameters that must be met for the Autostop option. Default is 0.01 (1%). *ACCMEAN* is ignored for *Astop* = ALL. The convergence of the mean values is met if for all random output parameters *y* the equation holds:

$$\frac{\left|\bar{y}(i)-\bar{y}(i-\mathrm{CHECK})\right|}{\bar{y}(i)} \le \mathrm{ACCMEAN} \qquad \text{with } i = 2\cdot\mathrm{CHECK}, 3\cdot\mathrm{CHECK}, \ldots$$

where the value of *CHECK* is specified in the *CHECK* option.

ACCSTDEV
Accuracy of the standard deviations of all random output parameters that must be met for the Autostop option. The default is 0.02 (2%). *ACCSTDEV* is ignored for *Astop* = ALL. The convergence for the standard deviations is met if for all random output parameters *y* the equation holds:

$$\frac{\left|\sigma_y(i)-\sigma_y(i-\mathrm{CHECK})\right|}{\sigma_y(i)} \le \mathrm{ACCSTDEV} \qquad \text{with } i = 2\cdot\mathrm{CHECK}, 3\cdot\mathrm{CHECK}, \ldots$$

where the value of *CHECK* is specified in the *CHECK* option.

CHECK
Sets how often conditions for convergence are checked for Autostop. The PDS checks if the convergence criteria are met every *i*-th loop, where *i* is given by the *CHECK* parameter. The default value is 10. It is not recommended to use *CHECK* = 1, because it could cause Autostop to terminate the simulations prematurely. The mean values and standard deviation might not show large differences between consecutive simulation loops but might still have a visible "global" trend if viewed over several simulations. This behavior indicates that convergence has not really been achieved. If you set *CHECK* = 1 (or another small number), then Autostop is not able to detect such a global trend. *CHECK* is ignored for *Astop* = ALL.

Seed
Seed value label. Random number generators require a seed value that is used to calculate the next random number. After each random number generation finishes, the seed value is updated and is used again to calculate the next random number. ANSYS initializes the seed value with the system time when the ANSYS session started.

CONT
Continues updating using the derived seed value (default).

TIME
Initializes the seed value with the system time. You can use this if you want the seed value set to a specific value for one analysis and then you want to continue with a "random" seed in the next analysis. It is not recommended to "randomize" the seed value with the *Seed* = TIME option for multiple analyses.

If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

INIT
: Initializes the seed value using 123457 (a typical recommendation). This option leads to identical random numbers for all random input variables when the exact analysis will be repeated, making it useful for benchmarking and validation purposes (where identical random numbers are desired). If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

Value
: Uses the specified (positive) value for the initialization of the seed value. This option has the same effect as *Seed* = INIT, except you can chose an arbitrary (positive) number for the initialization. If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

Command Default

NSIM = 30, *Astop* = AUTO, *ACCMEAN* = 0.01, *ACCSTDEV* = 0.02, *CHECK* = 10, *Seed* = CONT.

Notes

Defines the number of simulations, the specification of the Autostop option, how frequently the Autostop option is checked and the seed value for random number generation. If the Autostop option is enabled, then the number of simulations given for *NSIM* can be regarded as the maximum number of simulations to be performed. The Autostop option may terminate the simulations before *NSIM* simulations are done if the convergence criteria are met.

Menu Paths

Main Menu>Prob Design>Prob Method>Monte Carlo Sims

PDDOEL, *Name*, *Method*, *Vtype*, *Lopt*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*

Defines design of experiment levels for an individual random input variable.

PROBABILISTIC: Preprocessing

MP ME ST DY <> PR EM <> FL PP ED

Name
: Parameter name. The parameter name must have been previously defined as a random input variable using the **PDVAR** command.

Method
: Specifies the response surface method for which the levels of the design of experiment are to be defined. This field must not be left blank.

CCD
: Use the Central Composite Design method. The design experiment levels of a central composite design are defined in the fields *VAL1* to VAL5.

BBM
: Use the Box-Behnken Matrix method. The design experiment levels of a Box-Behnken Matrix design are defined in the fields *VAL1* to *VAL3*. The fields *VAL4* and *VAL5* are ignored

Vtype
Specifies the type of the values of the design of experiment levels.

PROB
The design of experiment levels are specified in terms of probabilities. This is the default.

PHYS
The design of experiment levels are specified in terms of physical values.

Lopt
Specifies the type of the design of experiment levels, indicating if they are defined by lower and upper bound only (default) or all specified by the user.

BND
You specify the lower and upper bounds for the design of experiment levels. The values for intermediate levels are calculated automatically at run time (default). The lower and upper levels of the design of experiment itself can be specified either in terms of probabilities or in terms of physical values, depending on the *Vtype* field.

For *Lopt* = BND and *Method* = CCD only the entries *VAL1* and VAL5 are processed and they represent the lower and upper bound values of a central composite design. The intermediate levels *VAL2* to *VAL4* are evaluated automatically. For *Lopt* = BND and *Method* = BBM only the entries *VAL1* and *VAL3* are processed and they represent the lower and upper bound values of a Box-Behnken Matrix design respectively. The intermediate level *VAL2* is evaluated automatically.

NOTE: The intermediate levels between the lower and upper bounds are calculated so they are at equal intervals along the curve (interpolated linearly in the physical space whether the curve is symmetrical or not) regardless of whether the lower and upper bounds are specified as probabilities (*Vtype* = PROB) or as physical values (*Vtype* = PHYS).

ALL
You explicitly specify all necessary design of experiment levels. The design of experiment levels can be in terms of probabilities or in terms of physical values, depending on the *Vtype* field.

VAL1, *VAL2*, *VAL3*, *VAL4*, *VAL5*
Values for the levels of the design of experiment for the random input variable *Name*. Must be specified in ascending order. All probabilities must be between 0.0 and 1.0.

For response surface methods based on a Central Composite Design (*Method* = CCD) *VAL1* through *VAL5* correspond to the five design of experiment levels of this method.

For response surface methods based on a Box-Behnken Matrix design (*Method* = BBM) *VAL1* through *VAL3* correspond to the three design of experiment levels of this method. The entries for *VAL4* and *VAL5* are ignored for *Method* = BBM.

Command Default

For *Meth* = CCD, *VAL1* = 0.005, *VAL2* = 0.05, *VAL3* = 0.5, *VAL4* = 0.95, *VAL5* = 0.995.

For *Meth* = BBM, *VAL1* = 0.01, *VAL2* = 0.5, *VAL3* = 0.99.

Notes

If *Vtype* = PHYS, you must enter values for *VAL1* through *VAL3* or *VAL5* (depending on the *Method* and *Lopt* option you choose). If *Vtype* = PROB and you do not enter values, they default to the values shown below.

For *Method* = CCD:

	Lopt = BND	*Lopt* = ALL
Vtype = PHYS	*VAL1* and *VAL5* must not be blank.	*VAL1* through *VAL5* must not be blank.
Vtype = PROB	*VAL1* = 0.005, *VAL5* = 0.995	*VAL1* = 0.005, *VAL2* = 0.05, *VAL3* = 0.50, *VAL4* = 0.95, *VAL5* = 0.995

For *Method* = BBM:

	Lopt = BND	*Lopt* = ALL
Vtype = Phys	*VAL1* and *VAL3* must not be blank.	*VAL1* through *VAL3* must not be blank.
Vtype = Prob	*VAL1* = 0.01, *VAL3* = 0.99	*VAL1* = 0.01, *VAL2* = 0.50, *VAL3* = 0.99

See Probabilistic Design in the *ANSYS Advanced Analysis Techniques Guide* for more information on the PDS methods.

Menu Paths

Main Menu>Prob Design>Prob Method>Response Surface

PDEF, *Lab*, *Item*, *Comp*, *Avglab*

Interpolates an item onto a path.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

Lab
Label assigned to the resulting path item (8 characters maximum). This item may be used as input for other path operations.

Item
Label identifying the item for interpolation. Valid item labels are shown in PDEF - Valid Item and Component Labels below. Some items also require a component label.

Comp
Component of the item (if required). Valid component labels are shown in PDEF - Valid Item and Component Labels below.

Avglab
Option to average across element boundaries:

AVG
Average element results across elements (default).

NOAV
Do not average element results across elements. If the parameter *DISCON* = MAT on the **PMAP** command, this option is automatically invoked.

Notes

Defines and interpolates a labeled path item along a predefined path [**PATH**]. Path item results are in the global Cartesian coordinate directions unless transformed [**RSYS**]. A path item must be defined before it can be used with other path operations. Additional path items may be defined from the **PVECT**, **PCALC**, **PDOT**, and **PCROSS** commands. Path items may be listed [**PRPATH**] or displayed [**PLPATH**, **PLPAGM**]. A maximum number of path items permitted is established by the *nSets* argument specified with the **PATH** command.

When you create the first path item [**PDEF** or **PVECT**], the program automatically interpolates four path items which are used to describe the geometry of the path. These predefined items are the position of the interpolated path points (labels XG, YG, and ZG) in global Cartesian coordinates, and the path length (label S). For alternate methods of mapping the path geometry (to include, for example, material discontinuity) see the **PMAP** command. These items may also be listed or displayed with the **PRPATH**, **PLPATH**, and **PLPAGM** commands.

The number of interpolation points on the path is defined by the *nDiv* argument on the **PATH** command. See the *ANSYS, Inc. Theory Reference* for details. Use **PDEF**,STAT to list the path item labels. Use **PDEF**,CLEAR to erase all labeled path items, except the path geometry items (XG, YG, ZG, S).

PDEF - Valid Item and Component Labels

Item	Comp	Description
	Valid item and component labels for nodal degree of freedom results are:	
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum.
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum.
TEMP[1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum.
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy (FLOTRAN).
ENDS		Turbulent energy dissipation (FLOTRAN).
	Valid item and component labels for element results are:	
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or Equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.

Item	Comp	Description
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
For contact results PowerGraphics is applicable for 3-D models only.		
CONT	STAT	Contact status. 3-closed and sticking, 2-closed and sliding, 1-open but near contact, 0-open and not near contact.
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
"	GAP	Contact gap distance.
"	FLUX	Total heat flux at contact surface.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component magnetic force or vector sum.
ETAB	Lab	Any user-defined element table label (see **ETABLE** command).
BFE	TEMP	Applied and calculated temperatures along a defined path.
		Valid item labels for FLOTRAN nodal results are:
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.

Item	Comp	Description
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
EVIS		Fluid effective viscosity.
CMUV		Turbulent viscosity coefficient.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.
SPHT		Specific heat.
RDFL		Radiation heat flux.

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, ..., TTOP instead of TEMP.

Menu Paths

Main Menu>General Postproc>Path Operations>Clear Path Items
Main Menu>General Postproc>Path Operations>Define Path>Path Status>Current Path
Main Menu>General Postproc>Path Operations>Map onto Path
Main Menu>General Postproc>Path Operations>Map onto Path>FE Results

---

PDEXE, *Slab*, *MRUN*, *NFAIL*, *FOPT*, *Fname*

Executes the probabilistic analysis.

PROBABILISTIC: Run Probabilistic Analysis

MP ME ST DY <> PR EM <> FL PP ED

Slab

Solution label. This is the name given to the files containing the results of different probabilistic analyses performed with a particular probabilistic model. Different solutions using different probabilistic methods or method options are stored in separate solution sets identified with this solution label. If the solution label is blank, ANSYS automatically assigns *Slab* = "SOLU*nn*", where *nn* is the current two-digit number of the probabilistic analysis (PDS allows a maximum of 10 solution sets identified by 10 solution labels); for example, the first analysis is SOLU01, the second is SOLU02, and so on up to SOLU10). This field cannot contain blanks. Maximum length of this field is 32 characters; if the field contains more than 32 characters, it will be truncated to 32 characters.

MRUN

Keyword for the processing of the simulation loops

SER

Executes the simulation loops of the probabilistic analysis using a single machine; serial processing (default).

PAR

Executes the simulation loops of the probabilistic analysis in parallel using the ANSYS parallel-processing tool. See Probabilistic Design in the *ANSYS Advanced Analysis Techniques Guide*.

NFAIL
This parameter sets the number of loops that are allowed to fail before the execution of the probabilistic analysis is terminated. For example, if geometric parameters are used as random variables then for some simulations (loops) the loop will fail because meshing was not successful. This might happen infrequently which might be acceptable. However, if the probabilistic problem is incorrectly defined then loops could fail frequently or even in every loop. You can define an upper limit for the number of failed loops that will be tolerated before the probabilistic analysis will be terminated. If you use *NFAIL* = 0 (default) then failed loops are not tolerated and the probabilistic analysis will terminate the first time a loop fails. *NFAIL* is used only if *MRUN* = PAR.

FOPT
Keyword for the file option. This determines if a copy of the sample file should be saved or not.

DEL
The sample file will be deleted (default).

COPY
A copy of the sample file will be saved as the file name specified with the Fname option.

Fname
Name of the copy of the sample file. The directory for storing the sample file is always the current working directory. The file extension is always **.csmp**. This option is ignored for *FOPT* = DEL.

Command Default

Slab = SOLU*nn* (see above) , *MRUN* = SER, *NFAIL* = 0, *FOPT* = DEL.

Notes

Runs the probabilistic analysis. The results of the probabilistic analysis are identified with the solution label specified here. This solution label is used for the postprocessing of the results. The maximum number of probabilistic analyses that can be executed, stored, and post-processed based on the same deterministic and probabilistic model in one session with the PDS is 10. However, it is not counted as a separate probabilistic analysis if the same solution label is used multiple times in order to append to or overwrite previous results.

The probabilistic results are stored in result files in ASCII format.

If you specify a solution label that has already been used, the results will be appended at the end of the results file.

After the **PDEXE** command has been issued, you can no longer change the probabilistic model using probabilistic preprocessing commands; doing so would invalidate the probabilistic analysis results and potentially corrupt the probabilistic database. Also, you are not allowed to change the probabilistic analysis file using a **PDANL** command after the first **PDEXE** has been executed.

If you need to change the probabilistic model (analysis file, random input variables, etc.) after the **PDEXE** command has been issued, you should use the **PDCLR**, POST command to clear the probabilistic results. We recommend that you first save the results using the **PDSAVE** command.

By default, ANSYS creates a sample file (called **Jobname.samp**) containing all values of all random input parameters; unless you specify otherwise, this file is deleted after the analysis is finished. With the *FOPT* and *Fname* options you can determine if and where a copy of that file should be stored for later use in another analysis.

Menu Paths

Main Menu>Prob Design>Run>Exec Serial >Run Serial

PDHIST, *Rlab*, *Name*, *NCL*, *Type*

Plots the frequency histogram.

PROBABILISTIC: Postprocessing

MP ME ST DY <> PR EM <> FL PP ED

Rlab

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an **RSFIT** command (for Response Surface Analyses).

Name

Parameter name. The parameter must have been previously defined as a random input variable or a random output parameter with the **PDVAR** command.

NCL

Number of classes for the histogram plot. This is the number of bars shown in the histogram. *NCL* must be a positive number. If this field is left blank, ANSYS calculates an appropriate number of classes based on the sample size. ANSYS divides the range between the smallest and largest sample value into *NCL* classes of equal width and determines the histogram frequencies by counting the number of hits that fall in the classes.

Type

Type of histogram.

ABS

Absolute frequency histogram. This is the actual number of hits in each class.

REL

Relative frequency histogram (default). This is the number of hits in the individual classes divided by the total number of samples.

NORM

Normalized frequency histogram. This is the number of hits in the individual classes divided by the total number of samples and divided by the width of the class. This normalization makes the histogram comparable to the probability density function.

Command Default

Rlab as described above, *NCL* as described above, *Type* = REL.

Notes

Plots the frequency histogram.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

The **PDHIST** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

Menu Paths

Main Menu>Prob Design>Prob Results>Statistics>Histogram

PDINQR, *Rpar*, *Name*, *Type*, *VAL*

Evaluates statistical characteristics of a random input variable.

PROBABILISTIC: Preprocessing

MP ME ST DY <> PR EM <> FL PP ED

Rpar
Name of scalar ANSYS parameter into which the characteristic value is stored.

Name
Parameter name. The parameter must have been previously defined as a random input variable using the **PDVAR** command.

Type
Keyword for the type of information you need to retrieve.

MEAN
Mean value of the random input variable specified in *Name*.

STDV
Standard deviation of the random input variable

PDF
Probability density function of the random input variable specified in *Name* at a given value specified by *VAL*.

CDF
Cumulative distribution function of the random input variable specified in *Name* at a given value specified by *VAL*.

INV
Inverse cumulative distribution function of the random input variable specified in *Name* at a given probability specified by *VAL*.

VAL
Value needed for *Type* = PDF, CDF, or INV. For *Type* = PDF and *Type* = CDF, this is the value of the random input variable at which the probabilistic density or cumulative distribution function should be evaluated. For *Type* = INV, *VAL* indicates the probability at which you want the inverse cumulative distribution function evaluated.

Notes

Evaluates statistical characteristics of a random input variable. The result is stored in the ANSYS parameter *Rpar*.

Menu Paths

Main Menu>Prob Design>Prob Definitns>Inquire

PDLHS, *NSIM*, *NREP*, *ISopt*, --, *Astop*, *ACCMEAN*, *ACCSTDV*, *CHECK*, *Seed*

Specifies options for Monte Carlo Simulations using Latin-Hypercube sampling.

PROBABILISTIC: Methods

MP ME ST DY <> PR EM <> FL PP ED

NSIM

Number of simulation loops per repetition cycle.

NREP

Number of repetition cycles of the analysis.

ISopt

Latin-Hypercube sampling divides the domain of each random input variable into intervals of equal probability. The interval sampling option *ISopt* determines where the samples are located within each interval.

RAND

Picks a random location within the interval (default).

MEAN

Picks the mean value location within the interval.

MEDI

Picks the median value location within the interval.

--

Unused field.

Astop

Autostop option label.

AUTO

Enable Autostop. When Autostop is used, the PDS feature continues the simulation loops until the convergence criteria for the mean value and the standard deviation have been met or until the number of simulations *NSIM* are complete, whichever comes first. The convergence criteria (mean value and standard deviations of all random output parameters) are specified by the *ACCMEAN* and *ACCSTDEV* parameters. The criteria are met if the mean value *and* the standard deviations converge within the accuracy specified in the *ACCMEAN* and *ACCSTDEV* options. The convergence check is done every *i*-th loop, where *i* is specified in the *CHECK* parameter.

ALL

Disable Autostop option. All Monte Carlo Simulations as specified by *NSIM* and *NREP* are performed (default).

ACCMEAN

Accuracy of the mean values of all random output parameters that must be met to activate Autostop. Default is 0.01 (1%). *ACCMEAN* is ignored for *Astop* = ALL. The convergence for the mean values is met if for all random output parameters *y* the following equation is true:

$$\frac{\left|\bar{y}(i)-\bar{y}(i-\mathrm{CHECK})\right|}{\bar{y}(i)} \le \mathrm{ACCMEAN} \qquad \text{with } i = 2\cdot\mathrm{CHECK}, 3\cdot\mathrm{CHECK}, \ldots$$

where the value of *CHECK* is given by the *CHECK* option.

ACCSTDEV

Accuracy of the standard deviations of all random output parameters that must be met to activate Autostop. The default is 0.02 (2%). *ACCSTDEV* is ignored for *Astop* = ALL. The convergence for the standard deviations is met if for all random output parameters *y* the following equation is true:

$$\frac{\left|\sigma_y(i)-\sigma_y(i-\text{CHECK})\right|}{\sigma_y(i)} \le \text{ACCSTDEV} \qquad \text{with } i = 2\cdot\text{CHECK}, 3\cdot\text{CHECK}, \ldots$$

where the value of *CHECK* is given by the *CHECK* option.

CHECK

Sets how often conditions for convergence are checked for Autostop. The PDS feature checks if the convergence criteria are met every *i*-th loop, where *i* is given by the *CHECK* parameter. The default value is 10. It not recommended to use *CHECK* = 1, because it could cause Autostop to terminate the simulations prematurely. The mean values and standard deviation might not show large differences between all simulation loops but might still have a visible "global" trend if viewed over several simulations. This behavior indicates that convergence has not really been achieved. If you set *CHECK* = 1, then Autostop is not able to detect such a global trend. *CHECK* is ignored for *Astop* = ALL.

Seed

Seed value label. Random number generators require a seed value that is used to calculate the next random number. After each random number generation finishes, the seed value is updated and is used again to calculate the next random number. ANSYS initializes the seed value with the system time when the ANSYS session started.

CONT

Continues updating using the derived seed value (default).

TIME

Initializes the seed value with the system time. You can use this if you want the seed value set to a specific value for one analysis and then you want to continue with a "random" seed in the next analysis. It is not recommended to "randomize" the seed value with the *Seed* = TIME option for multiple analyses. If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

INIT

Initializes the seed value using 123457 (a typical recommendation). This option leads to identical random numbers for all random input variables when the exact analysis will be repeated, making it useful for benchmarking and validation purposes (where identical random numbers are desired). If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

Value

Uses the specified (positive) value for the initialization of the seed value. This option has the same effect as *Seed* = INIT, except you can chose an arbitrary (positive) number for the initialization. If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

Command Default

NSIM = 30, *NREP* = 1, *ISopt* = RAND, *Astop* = ALL, *ACCMEAN* = 0.01, *ACCSTDEV* = 0.02, *CHECK* = 10, *Seed* = CONT.

Notes

Defines the number of simulations per repetition cycle, number of repetition cycles, specification of the Autostop option, checking frequency for the Autostop option, and the seed value for random number generation.

For Latin-Hypercube sampling, it is advantageous to divide the total number of requested simulations into a few repetitions. This adds more randomness to the sampling process. If NTOT is the total number of simulations, then as a rough rule of thumb NTOT should be *NREP* = $\sqrt[4]{\text{NTOT}}$ repetitions. The number obtained with this rule of thumb must be adjusted such that NTOT = *NREP***NSIM*. For example if NTOT = 1000 then *NREP* = $\sqrt[4]{1000}$ = 10, so the 1000 simulations can be done in 100 simulations with 10 repetitions. If for example NTOT = 100 then *NREP* = $\sqrt[4]{100}$ = 3.16, which means that the 100 simulations could be broken up into either 2*50 or 4*25 simulations.

If Autostop is enabled then the maximum number of simulations to be performed is given by *NSIM***NREP*. The Autostop option will terminate the simulations before the *NSIM***NREP* simulations are done if the convergence criteria are met.

Menu Paths

Main Menu>Prob Design>Prob Method>Monte Carlo Sims

PDMETH, *Method*, *Samp*

Specifies the probabilistic analysis method.

PROBABILISTIC: Methods

MP ME ST DY <> PR EM <> FL PP ED

Method

Label for the probabilistic analysis method.

MCS
: Monte Carlo Simulation

RSM
: Response Surface Method

Samp

Label for the sampling techniques. The sampling technique determines the values of the random input variables during the simulation loops.

DIR
: Direct or Crude Monte Carlo Sampling. This technique randomly samples the random input variables according to their distribution functions without "memory" of previous simulations.

  The parameters for a Monte Carlo Simulation using direct sampling are specified with the **PDDMCS** command.

LHS
: Latin Hypercube Sampling (default). Valid only for *Method* = MCS. For this sampling technique the random input variables are sampled randomly according to their distribution functions, efficiently stratifying the

samples into layers and avoiding the re-use of those layers. The sampling process has a "memory" of previous simulations, which prevents accumulation of clusters of samples. In addition, this sampling strategy forces the extreme ends of a distribution function to participate in the sampling. This generally leads to smoother distribution functions of the sampled set.

The parameters for a Monte Carlo Simulation using Latin-Hypercube sampling are specified with the **PDLHS** command.

USER
User specified sampling. Valid only for *Method* = MCS. In this case you provide a file containing the sampling "points" (values) of all random input variables for all simulation loops. These samples are simply executed and it is your responsibility to specify the samples correctly. The probabilistic design system can perform only limited checks on the samples you provide. ANSYS allows Monte Carlo specific post-processing operations on the results generated with user-specified samples. The parameters for the user-supplied sampling technique are specified with the **PDUSER** command.

CCD
Central Composite Design. Valid only for *Method* = RSM. A central composite design is composed of a center point, axis points, and corner points, called factorial points. Using large numbers of random input variables produces prohibitively large numbers of factorial points; therefore, ANSYS automatically reduces the number of factorial points by switching to a fractional plan for the factorial part of the design. See the **PDDOEL** command for more information.

Note — This option is only valid for 2 to 20 random input variables. You will receive an error if you have specified fewer than 2 or more than 20 random input variables.

BBM
Box-Behnken Matrix Design. Valid only for *Method* = RSM. A Box-Behnken Matrix design is composed of a center point plus the points at the middle of the edges of the hypercube in the space of random input variables. A Box-Behnken design might be advantageous if the corner points of the hypercube represent very extreme conditions that are undesirable and therefore should not be used for the sampling. See the **PDDOEL** command for more information.

Note — This option is only valid for 3 to 12 random input variables. You will receive an error if you have specified fewer than 3 or more than 12 random input variables.

USER
User specified sampling. In this case you provide a file containing the sampling "points" (values) of all random input variables for all simulation loops. These samples are simply executed and it is your responsibility to specify the samples correctly. The PDS can perform only limited checks on the samples you provide, if user-supplied sampling technique are specified with the **PDUSER** command.

Command Default

Method = MCS, *Samp* = LHS.

Notes

Specifies the probabilistic analysis method and the sampling technique used for the individual probabilistic analysis method.

Menu Paths

Main Menu>Prob Design>Prob Method>Monte Carlo Sims
Main Menu>Prob Design>Prob Method>Response Surface

PDOT, *LabR*, *LabX1*, *LabY1*, *LabZ1*, *LabX2*, *LabY2*, *LabZ2*

Calculates the dot product of two path vectors along the current path.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

LabR
Label assigned to dot product result.

LabX1
X-component of first vector label (labeled path item).

LabY1
Y-component of first vector label (labeled path item).

LabZ1
Z-component of first vector label (labeled path item).

LabX2
X-component of second vector label (labeled path item).

LabY2
Y-component of second vector label (labeled path item).

LabZ2
Z-component of second vector label (labeled path item).

Menu Paths

Main Menu>General Postproc>Path Operations>Dot Product

PDPINV, *Rlab*, *Name*, *PROB*, --, *CONF*

Prints the result of the inversion of a probability.

PROBABILISTIC: Postprocessing

MP ME ST DY <> PR EM <> FL PP ED

Rlab
Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an **RSFIT** command (for Response Surface Analyses).

Name
Parameter name. The parameter must have been previously defined as a random input variable or a random output parameter with the **PDVAR** command.

PROB
Target probability for which the random parameter value should be determined.

--
Unused field.

CONF
Confidence level. The confidence level is used to print the confidence bounds on the random parameter value. The value for the confidence level must be between 0.0 and 1.0 and it defaults to 0.95 (95%). Printing of confidence bound is suppressed for *CONF* $\leq$ 0.5. This parameter is ignored for response surface methods results postprocessing.

Command Default

Rlab as described above, *CONF* = 0.95.

Notes

Prints the value for the random parameter *Name* at which the probability that there are simulation values lower than that value is equal to *PROB*. This corresponds to an inversion of the cumulative distribution function (see **PDCDF** command) at a given probability. In this sense the **PDPINV** is doing the opposite of the **PDPROB** command. The **PDPROB** command evaluates a probability for a given random parameter value and the **PDPINV** command evaluates the random parameter value that corresponds to a given probability.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

The confidence level is a probability expressing the confidence that the value for the requested result is in fact between the confidence bounds. The larger the confidence level, the wider the confidence bounds. Printing the confidence bounds only makes sense for postprocessing Monte Carlo simulation results, where the confidence bounds represent the accuracy of the results. With increasing sample sizes, the width of the confidence bounds gets smaller for the same confidence level. For response surface analysis methods, the number of simulations done on the response surface is usually very large; therefore, the accuracy of the results is determined by the response surface fit and not by the confidence level.

The **PDPINV** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

Menu Paths

Main Menu>Prob Design>Prob Results>Statistics>Inverse Prob

PDPLOT, *Name*, *PLOW*, *PUP*

Plots the distribution curves of a defined random input variable.

PROBABILISTIC: Preprocessing

MP ME ST DY <> PR EM <> FL PP ED

Name
Parameter name. The parameter name must have been previously defined as a random input variable using the **PDVAR** command.

PLOW
Lower probability level used to determine the lower boundary of the curve. This probability must be between 0.0 and 1.0 and it defaults to 0.0025. This parameter is used to determine the lower plotting range (boundary) in case the random input variable does not have a minimum value (such as Gauss).

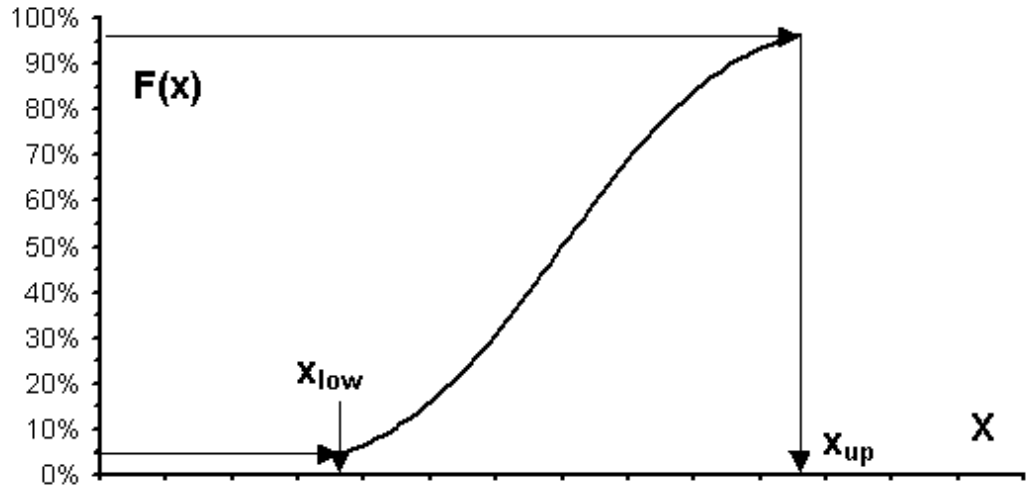
PUP
Upper probability level used to determine the upper boundary of the curve. This probability must be between 0.0 and 1.0 and it defaults to 0.9975.

Command Default

PLOW = 0.0025 (applied if distribution type has no minimum), *PUP* = 0.9975 (applied if distribution type has no maximum).

Notes

Plots the distribution of a defined random input variable. The **PDPLOT** command generates a probability density function plot as well as a cumulative distribution function plot of the random variable. The probabilities *PLOW* and *PUP* are used to determine the plot range of the random input variable values. To do this, the probabilities are converted into random input variable values using the inverse cumulative distribution function of the random input variable as shown in the following illustration.



Using the probabilities ensures that the boundaries are always feasible and meaningful for the random input variable regardless of its distribution type.

If *PLOW* is left blank, then a minimum value of the distribution is used for plotting, provided it exists (for example, uniform distribution). If the distribution type has no minimum value (for example, a Gaussian distribution) then the default value is used to determine the lower plotting range value. The same applies for the maximum value, if *PUP* is left blank.

Menu Paths

Main Menu>Prob Design>Prob Definitns>Plot

PDPROB, *Rlab*, *Name*, *Relation*, *LIMIT*, *--*, *CONF*

Prints a probability result.

PROBABILISTIC: Postprocessing

MP ME ST DY <> PR EM <> FL PP ED

Rlab
Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an **RSFIT** command (for Response Surface Analyses).

Name
Parameter name. The parameter must have been previously defined as a random input variable or a random output parameter using the **PDVAR** command.

Relation
Relation label for the relation between probabilistic design parameter *Name* and the limit value *LIMIT*:

LT
Less than (default).

GT
Greater than.

LIMIT
Limit value.

--
Unused field.

CONF
Confidence level. The confidence level is used to print the confidence bounds on the probability. The value for the confidence level must be between 0.0 and 1.0 and it defaults to 0.95 (95%). Printing of confidence bound is suppressed for *CONF* $\leq$ 0.5. This parameter is ignored for response surface methods results post-processing.

Command Default

Rlab as described above, *Relation* = LT, *CONF* = 0.95.

Notes

Prints the probability that the probabilistic design input or output variable denoted with *Name* is smaller or larger than a certain limit value.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

Use the relation label *Relation* to specify if you want to print a traditional probability value (LT) or the exceedence probability (GT). The *LIMIT* directly specifies at which value of *Name* (the design parameter) the probability should be evaluated. If *LIMIT* is between two sample values of *Name* the resulting probability is linearly interpolated between the sample values. If *LIMIT* is smaller than all sample values of *Name* the probability is 0.0 for *Relation* = LT and 1.0 for *Relation* = GT. If *LIMIT* is greater than all sample values for *Name* the probability is 1.0 for *Relation* = LT and 0.0 for *Relation* = GT.

The confidence level is a probability expressing the confidence that the value for the requested probability is in fact between the confidence bounds. The larger the confidence level, the wider the confidence bounds. Printing the confidence bounds only makes sense for postprocessing Monte Carlo simulation results, where the confidence bounds represent the accuracy of the results. With increasing sample sizes, the width of the confidence bounds gets smaller for the same confidence level. For response surface analysis methods, the number of simulations done on the response surface is usually very large; therefore, the accuracy of the results is determined by the response surface fit and not by the confidence level.

The **PDPROB** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

Menu Paths

Main Menu>Prob Design>Prob Results>Statistics>Probabilities

PDRAG, *PX1*, *PY1*, *PZ1*, *H1*, *PX2*, *PY2*, *PZ2*, *H2*, *Kcord*

Defines the external fluid drag loading for a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

PX1, *PY1*, *PZ1*
External fluid drag pressure (global Cartesian components) at height *H1*.

H1
Height (along *Kcord* coordinate) for first drag pressure.

PX2, *PY2*, *PZ2*
External fluid drag pressure (global Cartesian components) at height *H2*.

H2
Height (along *Kcord* coordinate) for second drag pressure.

Kcord
Coordinate direction for height value (in the global Cartesian coordinate system):

X
X coordinate.

Y
Y coordinate (default).

Z
Z coordinate.

Notes

Defines the external fluid drag loading (pressure) as a function of height for a piping run. See the PREP7 **RUN** command. The element drag pressure is determined from the centroid height and linear interpolation. Pressures are assigned to the elements as they are generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Loads

PDRESU, *Fname*, *Ext*, --

Reads the probabilistic model data and loads it into the database.

PROBABILISTIC: Database

MP ME ST DY <> PR EM <> FL PP ED

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext

Filename extension (8 character maximum).

The extension defaults to pds if *Fname* is blank; otherwise, no default.

--

Unused field

Command Default

Fname = jobname, *Ext* = **pds**, the default directory is the current working directory.

Notes

Reads the probabilistic model data from the specified file and loads it into the database. Probabilistic analyses results are not stored in the database with the **PDRESU** command, rather they reside in separate results files. Analyses results are loaded automatically (one-by-one and on demand) when a probabilistic postprocessing command is issued.

Menu Paths

Main Menu>Prob Design>Prob Database>Resume

PDROPT, *RVAR*, *CORR*, *STAT*, *SHIS*, *HIST*, *CDF*, *SENS*, *CMAT*, *CONF*

Specifies the options for an HTML report.

PROBABILISTIC: Postprocessing

MP ME ST DY <> PR EM <> FL PP ED

RVAR

Specifies in which form to show the definitions of random variables in the report.

0

Using tables (including name and distribution parameter) and figures (including a probability density function plot and a cumulative distribution plot) (default).

1

Using tables only.

2

Using figures only.

3
None.

CORR
Specifies if a table describing the correlation between random variables should be included in the report.

0
Yes, include this table (default).

1
No, do not include this table.

STAT
Specifies which statistics to include in the report. In general, statistics are provided in a tabular form.

0
Statistics of the random output parameters only (default).

1
Statistics of the random input variables only.

2
Statistics of both the random input variables and the random output parameters.

3
None.

SHIS
Specifies which sample history plots to include in the report. This option applies to the random output parameters only.

0
None (default).

1
Mean value and standard deviation as a sample plot.

2
Mean value, standard deviation and sample values as a sample plot.

3
All types of sample plots - mean value, standard deviation, minimum value, maximum values, and the sample values.

HIST
Specifies which histogram plots to include in the report.

0
Histogram of the random output parameters only (default).

1
Histogram of the random input variables only.

2
Histogram of both the random input variables and the random output parameters.

3
None.

CDF

Specifies which cumulative distribution function (CDF) plots to include in the report.

0
CDF of the random output parameters only (default).

1
CDF of the random input variables only.

2
CDF of both the random input variables and the random output parameters.

3
None.

SENS

Specifies which sensitivity plots to include in the report.

0
Plots the sensitivities for all random output parameters based on Spearman-rank-order correlation coefficient (default).

1
Plots the sensitivities for all random output parameters based on linear (Pearson) correlation coefficient.

2
Plots the sensitivities according to option *SENS*=1 and *SENS*=2.

3
None.

CMAT

Specifies which correlation matrices to include in the report.

0
Correlation matrix between random output parameters and random output parameters only (default).

1
Correlation matrix between random input variables and random output parameters only.

2
Correlation matrix between random input variables and random input variables only.

3
Correlation matrices according to option *CMAT*=0 and *CMAT*=1.

4
Correlation matrices according to option *CMAT*=0 and *CMAT*=2.

5
Correlation matrices according to option *CMAT*=1 and *CMAT*=2.

6
Correlation matrices according to option *CMAT*=0, *CMAT*=1, and *CMAT*=2.

7
None.

CONF
Confidence level. The confidence level is used to plot confidence bounds for the history value. The value for the confidence level must be between 0.0 and 1.0 and it defaults to 0.95 (95%). Confidence bound(s) plotting is suppressed for *CONF* $\leq$ 0.5. This option is ignored if the report does not include plots for which confidence bounds are applicable.

Command Default

RVAR = 0, *CORR* = 0, *STAT* = 0, *SHIS* = 0, *HIST* = 0, *CDF* = 0, *SENS* = 0, *CMAT* = 0, *CONF* = 0.95.

Notes

Specifies the options for an HTML report. An HTML report includes a description of the deterministic model, the probabilistic model, the probabilistic methods used for the analyses, and the results obtained from the analyses. The deterministic model is documented in the report by including a link to the analysis file (see **PDANL** command). In addition, an element plot of the component is shown, if available, based on the current view settings. The command **ALLSEL** is issued automatically prior to the respective plot command.

Menu Paths

Main Menu>Prob Design>Prob Results>Report>Report Options

/PDS

Enters the probabilistic design system.

PROBABILISTIC: Auxiliary

MP ME ST DY <> PR EM <> FL PP ED

Notes

Enters the Probabilistic Design System (PDS). This command is valid only at the Begin Level.

Menu Paths

Main Menu>Prob Design

PDSAVE, *Fname*, *Ext*, --

Writes the probabilistic model data to a file.

PROBABILISTIC: Database

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to pds if *Fname* is blank; otherwise, no default.

--
Unused field

Command Default

Fname = **Jobname**, *Ext* = **pds**, the default directory is the current working directory.

Notes

Writes the probabilistic model data to a file. Saved data include probabilistic data only; the results of the probabilistic analyses are not stored in the file (rather, these are stored in separate result files).

Menu Paths

Main Menu>Prob Design>Prob Database>Save

PDSCAT, *Rlab*, *Name1*, *Name2*, *Type*, *ORDER*, *NMAX*

Plots a scatter graph.

PROBABILISTIC: Postprocessing

MP ME ST DY <> PR EM <> FL PP ED

Rlab
Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command, or the response surface set label defined in an **RSFIT** command.

Name1, *Name2*
Parameter names. The parameters must have been previously defined as a random input variable or a random output parameter using the **PDVAR** command. The parameter data for *Name1* is shown on the X-axis and the parameter data for *Name2* is shown on the Y-axis in the plot.

Type
Keyword for the type of trendline curve.

POLY
Polynomial trendline (default).

NONE
A trendline is not plotted.

ORDER
Order of the polynomial trendline. This parameter is used only for *Type* = POLY. *ORDER* must be a positive number. There is no maximum for *ORDER* provided there are enough data points to evaluate a polynomial of the requested order. Default is 1.

NMAX
Maximum number of points plotted in the scatter plot. If there are more sample data, then only the first *NMAX* points are plotted. The default value is 10,000.

Command Default

Rlab as described above, *Type* = POLY, *ORDER* = 1, *NMAX* = 10,000

Notes

Plots a scatter graph with or without a trendline. The scatter plot shows the simulated points for two random parameters. Random input variables and random output parameters are valid for both X- and Y-axis. The mean value of both parameters are marked with separate green lines. The point where the green lines cross marks the statistical center of gravity of the cloud of all simulated data points of the two parameters.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

If the parameter data for *Name1* includes negative values, fitting a logarithmic trendline is not possible and the logarithmic trendline plot is suppressed if requested. The same applies for an exponential trendline if the data for the *Name2* includes negative values.

Because of the amount of data involved, the scatter plot is limited to *NMAX* points. If shown, the trendline is evaluated only on the *NMAX* points shown. However, the statistical information shown in the plot, such as the mean value lines for both parameters and the correlation coefficients listed in the legend are based on the full set of samples. If required, you can increase *NMAX* to plot more points, but this will affect the time needed to process the **PDSCAT** command. If *NMAX* is less than the total amount of simulated points, which is typically possible for Response Surface Methods, you will see an appropriate warning in the plot legend.

Menu Paths

Main Menu>Prob Design>Prob Results>Trends>Scatter Plot

PDSENS, *Rlab*, *Name*, *Chart*, *Type*, *SLEVEL*

Plots the probabilistic sensitivities.

PROBABILISTIC: Postprocessing

MP ME ST DY <> PR EM <> FL PP ED

Rlab

Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command (if you are directly postprocessing Monte Carlo Simulation results), or the response surface set label defined in an **RSFIT** command (for Response Surface Analyses). The **PDSENS** command cannot be used to postprocess the results in a solution set that is based on Response Surface Methods, only Monte Carlo Simulations.

Name

Parameter name. The parameter must have been previously defined as a random output parameter using the **PDVAR** command.

Chart

Keyword for the type of chart to be plotted.

BAR

Bar chart of the absolute sensitivities.

PIE
Pie chart of relative and normalized sensitivities.

BOTH
Both pie and bar charts plotted side by side (default).

Type
Keyword for the type of correlation coefficients used to evaluate the sensitivities.

RANK
Spearman rank-order correlation coefficient (default).

LIN
Pearson linear correlation coefficient.

SLEVEL
Significance level. The value for the significance level must be between 0.0 and 1.0 and it defaults to 0.025 (2.5%).

Command Default

Rlab as described above, *Chart* = BOTH, *Corr* = RANK, *SLEVEL* = 0.025

Notes

Plots the probabilistic sensitivities.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

Evaluation of the probabilistic sensitivities is based on the correlation coefficients between all random input variables and the random output parameter specified by *Name*. You can chose which correlation coefficient should be used for that evaluation using the *Corr* option. For all sensitivity values, the probabilistic design tool evaluates the probability that the sensitivity can be neglected, based on statistical test theory. If this probability exceeds the significance level as specified by the *SLEVEL* parameter, the sensitivity value should be regarded as negligible or insignificant. The higher the significance level (*SLEVEL*) the more sensitivities are considered as significant. The sensitivity plot includes the significant sensitivities only and lists the insignificant ones separately.

Menu Paths

Main Menu>Prob Design>Prob Results>Trends>Sensitivities

PDSHIS, *Rlab*, *Name*, *Type*, *CONF*
Plots the sample history values.

PROBABILISTIC: Postprocessing

MP ME ST DY <> PR EM <> FL PP ED

Rlab
Result set label. Identifies the result set to be used for postprocessing. A result set label can be the solution set label you defined in a **PDEXE** command, or the response surface set label defined in an **RSFIT** command.

Name
Parameter name. The parameter must have been previously defined as a random input variable or a random output parameter using the **PDVAR** command.

Type
Keyword to identify which type of data is to be plotted:

SAMP
Sampled values (default).

MEAN
Mean values for results based on Monte Carlo simulation methods only.

STDV
Standard deviations for results based on Monte Carlo simulation methods only.

MIN
Minimum values for results based on Monte Carlo simulation methods only.

MAX
Maximum values for results based on Monte Carlo simulation methods only.

MEAN, STDV, MIN, and MAX are only valid for Monte Carlo methods.

CONF
Confidence level. The confidence level is used to plot confidence bounds for the history value. The value for the confidence level must be between 0.0 and 1.0 and it defaults to 0.95 (95%). Confidence bound(s) plotting is suppressed for *CONF* $\leq$ 0.5. This option is ignored for *Type* = SAMP (no confidence bounds are plotted).

Command Default

Rlab as described above, *Type* = SAMP, *CONF* = 0.95

Notes

Plots the sample history values as a function of the number of simulation loops.

If *Rlab* is left blank, then the result set label is inherited from the last **PDEXE** command (*Slab*), **RSFIT** command (*RSlab*), or the most recently used PDS postprocessing command where a result set label was explicitly specified.

The confidence level is a probability expressing the confidence that the value for the requested probability is in fact between the confidence bounds. The larger the confidence level, the wider the confidence bounds. For *Type* = MEAN and *Type* = STDV, lower and upper confidence curves are plotted. For *Type* = MEAN, the mean value curve starts at the first simulation and the confidence bounds start with simulation number 2. For *Type* = MIN only the upper confidence bound is shown (the interpretation is that by a certain probability the true minimum value is below this curve). This probability (or confidence) is set using *CONF*. Likewise, for *Type* = MAX, only the lower confidence bound is shown. For all *Type* options, confidence curves are plotted starting with the simulation at which enough data is available to calculate the bounds. However, for scaling reasons, no confidence bounds are plotted for simulation numbers 1 to 5 even if they might be available.

Menu Paths

Main Menu>Prob Design>Prob Results>Statistics>Sampl History

PDUSER, *Fname*, *Ext*, --

Specifies options for user-specified sampling methods.

PROBABILISTIC: Methods

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Ext
Filename extension (8 character maximum).

--
Unused field

Notes

If user-specified sampling methods are requested with the **PDMETH**, MCS, USER command or the **PDMETH**, RSM, USER command, then you need to specify which file contains the sample data. For more information on the format of this file, see Probabilistic Design in the *ANSYS Advanced Analysis Techniques Guide*.

Menu Paths

Main Menu>Prob Design>Prob Method>Monte Carlo Sims
Main Menu>Prob Design>Prob Method>Response Surface

PDVAR, *Name*, *Type*, *PAR1*, *PAR2*, *PAR3*, *PAR4*

Specifies the parameters to be treated as probabilistic design variables.

PROBABILISTIC: Preprocessing

MP ME ST DY <> PR EM <> FL PP ED

Name
Parameter name (must be a scalar ANSYS parameter). The parameter must have been previously defined as a random input variable or a random output parameter with the **PDVAR** command. See the ***SET** command for restrictions about ANSYS parameters.

Type
Probabilistic design variable type. This is the statistical distribution type. For more information on each of these types, see Probabilistic Design in the *ANSYS Advanced Analysis Techniques Guide*.

BETA
Beta distributed random variable.

PAR1 = Shape parameter. Defaults to 2.0.

PAR2 = Shape parameter. Defaults to 2.0.

PAR3 = Lower minimum value. Defaults to 0.0.

PAR4 = Upper maximum value. Defaults to 1.0.

EXPO
Exponential distributed random variable.

PAR1 = Decay parameter λ. Must be larger then 0.0 and defaults to 1.0.

PAR2 = Shift or minimum value. Defaults to 0.0.

PAR3, *PAR4* are ignored.

GAMA
Gamma distributed random variable.

PAR1 = Decay parameter λ. Must be larger then 0.0 and defaults to 1.0.

PAR2 = Exponential parameter k. Must be larger then 0.0 and defaults to 1.0.

PAR3, *PAR4* are ignored. Exponential distributed random variable.

GAUS
Gaussian (Normal) distributed random variable.

PAR1 = Mean value. Defaults to 0.0.

PAR2 = Standard deviation. Must be larger then 0.0 and defaults to 1.0.

PAR3, *PAR4* are ignored.

LOG1
Lognormal distributed random variable specified directly with the statistical parameters mean value and standard deviation.

PAR1 = Mean value. Must be larger then 0.0 and defaults to 1.0.

PAR2 = Standard deviation. Must be larger then 0.0 and defaults to 1.0.

PAR3, *PAR4* are ignored.

LOG2
Lognormal distributed random variable specified with the statistical parameters mean value and standard deviation of the logarithm of the random values.

PAR1 and *PAR2* must also be defined. *PAR1* = Mean value of the logarithm of the data. Defaults to 0.0.

PAR2 = Standard deviation of the logarithm of the data. Must be larger then 0.0 and defaults to 1.0.

PAR3, *PAR4* are ignored.

UNIF
Uniform distributed random variable. Note that *PAR1* must be less than *PAR2*.

PAR1 = Minimum value. Defaults to 0.0.

PAR2 = Maximum value. Defaults to 1.0.

PAR3, *PAR4* are ignored.

TGAU
Truncated Gaussian distributed random variable. Note that *PAR3* must be less than *PAR4*.

PAR1 = Mean value of the untruncated Gaussian distribution. Defaults to 0.0.

PAR2 = Standard deviation of the untruncated Gaussian distribution. Must be larger then 0.0 and defaults to 1.0.

PAR3 = Minimum value and lower truncation boundary. Defaults to -3.0.

PAR4 = Maximum value and upper truncation boundary. Defaults to +3.0.

TRIA
Triangular distributed random variable. Note that *PAR1* must be less than *PAR2* which must be less than *PAR3*.

PAR1 = Minimum value. Defaults to 0.0.

PAR2 = Most Likely Value (MLV). Defaults to 0.5.

PAR3 = Maximum value. Defaults to 1.0.

PAR4 is ignored.

WEIB
Weibull (Type III smallest) distributed random variable. Note that *PAR2* must be greater than *PAR3*. If *PAR3* = 0.0, the random distribution is equivalent to a two-parameter Weibull distribution.

PAR1 = Weibull exponent. Must be larger then 0.0 and defaults to 1.0.

PAR2 = Characteristic value. Must be larger then 0.0 and defaults to 1.0.

PAR3 = Shift or minimum value. Defaults to 0.0.

PAR4 is ignored.

RESP
Random output or response parameter.*PAR1* to *PAR4* are not used.

DEL
Deletes this probabilistic design variable (does not delete the ANSYS parameter). This option is only valid if the parameter *Name* was previously defined as a probabilistic design variable (using *Type* = BETA, ..., WEIB or *Type* = RESP). The parameter retains the value assigned during the last probabilistic design loop.*PAR1* to *PAR4* are not used.

PAR1, ..., PAR4
Parameters of the distribution function. The parameters must be specified according to the requirements of the individual distribution types described above.

Notes

Specifies the parameters to be treated as probabilistic design variables. A random input variable is specified by the name of the ANSYS parameter, the type of the distribution function (*Type*) and its distribution parameters (*PAR1*, ..., *PAR4*). A random output parameter is specified by the name of the ANSYS parameter and the type identifying it as a random output parameter (*Type* = RESP).

Menu Paths

Main Menu>Prob Design>Random Input

PDWRITE, *File*, *Fnam*, *Lnam*

Generates an HTML report for the probabilistic analyses.

PROBABILISTIC: Postprocessing

MP ME ST DY <> PR EM <> FL PP ED

File
File name and directory path (248 characters maximum, including directory) from which to read the report. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Fnam
First name of the author of the report (32 characters maximum). This first name must not include blanks.

Lnam
Last name of the author of the report (32 characters maximum). This last name must not include blanks.

Notes

Generates an HTML report for the probabilistic analysis. An HTML report includes a description of the deterministic model, the probabilistic model, the probabilistic methods used for the analyses and the results obtained from the analyses.

Menu Paths

Main Menu>Prob Design>Prob Results>Report>Generate Report

PEMOPTS, *TOLER*, *Method*

Defines percentage tolerance and error estimation method for electrostatic p-Method solution.

SOLUTION: Controls

MP ME ST <> <> <> EM <> <> PP ED

TOLER
Percentage tolerance used to determine which elements to fix at the current p-level. If an element's error energy is less that this tolerance, it's p-level will be fixed at the current level. Defaults to 5%.

Method
Specifies the error estimation method.

ZZ
Zienkiewicz-Zhu error estimator. Applicable to problems with a single dielectric material (default).

DUAL
Dual field method. Applicable to problems with single or multiple dielectric materials.

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>p-Method>Solution Options
Main Menu>Solution>Load Step Opts>p-Method>Solution Options

PERBC2D, *LOC1*, *LOC2*, *LOCTOL*, *R1*, *R2*, *TOLR*, *OPT*, *PLNOPT*

Generates periodic constraints for 2-D planar magnetic field analyses.

PREP7: Special Purpose

MP ME ST DY <> <> EM <> <> PP ED

LOC1
Constant coordinate location of the first plane of nodes. For *PLNOPT* = 1 or 2, the constant coordinate location is the global Cartesian coordinate system [**CSYS**,0] location in the X or Y direction respectively. For *PLNOPT* = 0, the location is the angle in the global cylindrical coordinate system [**CSYS**,1].

LOC2
Constant coordinate location of the second plane of nodes. For *PLNOPT* = 1 or 2, the constant coordinate location is the global Cartesian coordinate system [**CSYS**,0] location in the X or Y direction respectively. For *PLNOPT* = 0, the location is the angle (in degrees) in the global cylindrical coordinate system [**CSYS**,1].

LOCTOL
Tolerance on the constant coordinate location for node selection. Defaults to .00001 for *PLNOPT* = 1 or 2 and .001 degrees for *PLNOPT* = 0.

R1
Minimum coordinate location along the second plane of nodes. For *PLNOPT* = 1 or 2, the coordinate location is the global Cartesian coordinate system location in the Y or X direction respectively. For *PLNOPT* = 0, the coordinate location is the radial coordinate value in the global cylindrical coordinate system. Periodic conditions are not applied to nodes at this location.

R2
Maximum coordinate location along the second plane of nodes. For *PLNOPT* = 1 or 2, the coordinate location is the global Cartesian coordinate system location in the Y or X direction respectively. For *PLNOPT* = 0, the coordinate location is the radial coordinate value in the global cylindrical coordinate system. Periodic conditions are not applied to nodes at this location.

TOLR
Tolerance dimension on node selection along the plane of nodes. Defaults to .00001.

OPT
Periodic option:

0
Odd symmetry (default). Apply constraint equations such that AZ(i) = -AZ(j).

1
Even symmetry. Apply node coupling such that AZ(i) = AZ(j).

PLNOPT
Symmetry plane option:

0
Planes of constant angle in the global cylindrical coordinate system [**CSYS**,1].

1
Planes parallel to the global Cartesian X axis [**CSYS**,0].

2
Planes parallel to the global Cartesian Y axis [**CSYS**,0].

Notes

PERBC2D invokes an ANSYS macro which generates periodic boundary condition constraints for 2-D planar magnetic field analysis. The macro is restricted to node pairs sharing common coordinate values along symmetry planes separated by a constant coordinate value. Planes (or lines) must lie at either constant angles (*PLNOPT* = 0), constant X values (*PLNOPT* = 1), or constant Y values (*PLNOPT* = 2). **PERBC2D** applies constraint equations (*OPT* = 0, odd symmetry) or node coupling (*OPT* = 1, even symmetry) to each node pair sharing a common coordinate value along the symmetry planes. By default, periodic conditions are not applied at the first and last node pairs on the symmetry planes unless the input location values, R1 and R2, are adjusted to be less than or greater than the actual node coordinate values. Nodes are selected for application of the constraints using the **NSEL** command with tolerances on the constant coordinate location (*LOCTOL*) and the coordinate location along the plane (RTOL).

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Periodic BCs
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Periodic BCs

---

PERI, *DX*, *DY*, *DZ*

Specifies periodic boundary conditions in an incompressible flow analysis.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

DX
Offset in the X direction of the second boundary from the first.

DY
Offset in the Y direction of the second boundary from the first.

DZ
Offset in the Z direction of the second boundary from the first.

Notes

Specified offsets must be consistent with the flow field coordinate system, as dictated by the FLUID141 and FLUID142 KEYOPT(3) setting.

The meshes at the two boundaries must be identical.

Menu Paths

This command cannot be accessed from a menu.

PEXCLUDE, *ELEM*

Specifies elements to be excluded from p-level escalations.

SOLUTION: p-Method

MP ME ST <> <> PR <> <> <> PP ED

ELEM

Element number of element to be excluded. If *ELEM* = ALL, exclude all elements. If *ELEM* = STAT, give status of excluded elements. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

Command Default

No elements are excluded.

Notes

The elements chosen by this command will be excluded from convergence calculations. As a result, these elements will not have their p-levels increased throughout the solution iterations.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>p-Method>Stored Energy>Exclude Elems
Main Menu>Preprocessor>Loads>Load Step Opts>p-Method>Strain Energy>Exclude Elems
Main Menu>Solution>Load Step Opts>p-Method>Stored Energy>Exclude Elems
Main Menu>Solution>Load Step Opts>p-Method>Strain Energy>Exclude Elems

---

PFACT, *TBLNO*, *Excit*, *Parcor*

Calculates participation factors for the PSD or multi-point response spectrum table.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

TBLNO

Input PSD (Power Spectral Density) table number for which participation factors are to be calculated.

Excit

Label defining the location of excitation:

BASE

Base excitation (default).

NODE

Nodal excitation.

Parcor
Label defining excitation type (applies only to **SPOPT**,PSD analysis). Used only when partially correlated excitation is due to wave propagation or spatial correlation. Defaults to partially correlated excitation as defined by **COVAL** and **QDVAL** commands.

WAVE
Excitation defined by **PSDWAV** command.

SPAT
Excitation defined by **PSDSPL** command.

Notes

Calculates the participation factors for a particular PSD or multi-point response spectrum table defined with the **PSDVAL** command. The **Jobname.DB** file must contain modal solution data in order for this command to calculate the participation factor. There must be a **PFACT** command for each excitation spectrum.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Calculate PF
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Calculate PF
Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Calculate PF
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Calculate PF

PFLUID, *DENS*

Defines the contained fluid density for a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

DENS
Density of the contained fluid.

Notes

See the PREP7 **RUN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Specifications

---

PGAP, *NLOC*, *K*, *DX*, *DY*, *DZ*, *GAP*, *ELEM*

Defines a spring-gap constraint in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

NLOC
Node where gap is to be placed. Defaults to current run starting point.

K
Spring constant value (must be positive).

DX, *DY*, *DZ*
Increment (in terms of the active coordinate system components) to determine gap ground point. Element length must not be zero. Constraints are automatically generated at the ground point.

GAP
Gap size (defaults to the element length).

ELEM
Element number to be assigned to gap (defaults to MAXEL + 1).

Notes

Defines a spring-gap constraint (gap element, CONTAC52) at a given location in a piping run. Gives spring constraint resistance after a specified gap is closed. See the **RUN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Spring-Gap Supp

---

PGRAPH, *Option*, *Fname*, *Fext*, --

Specifies the location from which graphics data will be retrieved for viewing.

SOLUTION: Misc Loads

MP ME ST DY <> PR EM <> FL PP ED

Option
Switch that controls the PGR file write operations :

Off
Use the data currently in the data base for display (default).

On
Use the PowerGraphics data in the PGR file for display.

Fname
File name and directory path (248 characters maximum, including directory) from which to read the PGR file. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to *Jobname*, or *File* if no jobname is specified.

Fext
Filename extension (8 character maximum).

--
Unused field

Notes

Menu Paths

Main Menu>Solution>Output Controls>PGR File
Main Menu>General Postproc>Write PGR File

PGRSET, *Lstep*, *SBSTEP*, *--*, *KIMG*, *TIME*, *--*, *NSET*

Defines the data set to be read from the PGR file.

POST1: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Lstep
Load step number of the data set to be read (defaults to 1):

N
Read load step *N*.

FIRST
Store the first data set on the PGR file (*SBSTEP* and *TIME* are ignored).

LAST
Store the last data set on the PGR file. (*SBSTEP* and *TIME* are ignored).

NEXT
Store the next data set on the PGR file (*SBSTEP* and *TIME* are ignored). If at the last data set, the first data set will be read as the next.

LIST
Scan the PGR file and list a summary of each load step. (*FACT*, *KIMG*, *TIME* and *ANGLE* are ignored.)

SBSTEP
Substep number (within *Lstep*). If *Lstep* = LIST, *SBSTEP* = 0 or 1 lists the basic step information, whereas *SBSTEP* = 2 also lists the load step title, and labels imaginary data sets if they exist. Default maximum is 1000. When the number of substeps exceeds this limit, you need to issue **SET**,*Lstep*,LAST to bring in the 1000th load step. Use **/CONFIG** to increase the limit.

--
Unused field

KIMG
Used only with results from complex analyses.

0
Store real part of complex solution.

1
Store imaginary part.

TIME
Time-point identifying the data set to be read. For the harmonic response analyses, time corresponds to the frequency. If both *Lstep* and *SBSTEP* are zero (or blank), read data set at time = *TIME*. If *TIME* is beyond the last time point on the file, the last time point will be used. If *TIME* doe not match a time value on the PGR file, the nearest time point in the data set will be used.

--
Unused field

NSET
Data set number of the data set to be read. If a positive value for *NSET* is entered, *Lstep*, *SBSTEP*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by **PGRSET**,LIST.

Notes

Defines the data set to be read from the PGR file into the database. Various operations may also be performed during the read operation.

Menu Paths

Main Menu>Solution>Output Ctrls>PGR File
Main Menu>General Postproc>Write PGR File

PGSAVE, *Fname*, *Fext*, *--*, *DataType*, *InteriorKey*, *Append*

Creates a PowerGraphics (PGR) file from results data.

SOLUTION: Misc Loads

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to *Jobname*, or *File* if no jobname is specified.

Fext
Filename extension (8 character maximum).

--
Unused field

DataType
Data type to create. This specification applies to discontinuous results data such as stress, strain, and field data. .

0
Save as nodal-averaged data (default). Used by the **PLNSOL** command.

1
Reserved.

2
Save both nodal averaged and unaveraged data. Used by the **PLESOL** and **PLNSOL** commands.

InteriorKey
Key that controls whether or not internal model data is saved to the PGR file. Internal data is required in order to use graphical slicing, capping, vector displays, or Isosurface displays for data display. Interior data is also required for the **AVRES**,,FULL option, where surface results take into consideration the interior element contributions.

0
Save exterior surface data only. (default)

1
Save exterior and interior data.

Append
Append or overwrite data to the specified.

0
Append the data to the existing file (default). Note that the geometry must not have been changed between the write operations.

1
Overwrite the data file.

Notes

The user may elect to write results to the PGR file in a specified order. The specifications (including **/ESHAPE** parameters) must be defined before saving the file. The GUI will allow the user to write the current set, or all sets from the results file.

For contact analysis, only flex-flex analysis is supported.

Menu Paths

Main Menu>Solution>Output Ctrls>PGR File
Main Menu>General Postproc>Write PGR File

PGSELE, *Type*, *Item*, --, *VMIN*, *VMAX*, *VINC*

Select a subset of elements for display with the PGR viewer.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying type of select

S
Select a new set (default)

R
Reselect a set from the current set

A
Additionally select a set, extending the current set.

U
Unselect a set from the current set.

ALL
Restore the full set. Subsequent command arguments are ignored.

INVE
Invert the current set (change selected to unselected; change unselected to selected). Subsequent command arguments are ignored.

STAT
Display the current select status. Subsequent command arguments are ignored.

Item
Label identifying the data. Valid labels are:

If *Type* = STAT and *Item* = (blank) or ELEM, the element selection information is provided.
Type - Element type number
Mat - Material number
Real - Real constant number
Sec - Section identification number
Comp or CM - Component name(s). (see notes, below)
Pick - Pick entities via a picking window. (see notes, below)
(Defaults to Type).

--
Unused field

VMIN
Minimum value of *Item* range. Ranges are attribute numbers.

VMAX
Maximum value for *Item* range. VMAX defaults to VMIN for input values.

VINC
Value increment within *Item* range. Used only with integer ranges (such as for attribute numbers). Defaults to 1. VINC cannot be negative.

Command Default

All elements are selected.

Notes

When *Item* = Pick, interactive picking is enabled. The displayed facets of the elements you pick will be flagged according to how they are resolved in PowerGraphics. Exterior elements will be flagged on each facet surface, while elements with no external (surface) component will be flagged at the centroid of that element.

When *Item* = Comp, the subsequent values for *VMIN*, *VMAX*, and *VINC*, are replaced by valid component names (32 character max). You can name up to 16 components in this fashion. All named components must contain elements.

This command is valid in POST1

Menu Paths

Main Menu>General Postproc>Results Viewer

PGWRITE, *Label*, *Fname*, *Fext*, --, *DataType*, *InteriorKey*, *Append*

Writes selected solution data to the PGR file for faster post processing access.

SOLUTION: Misc Loads

MP ME ST DY <> PR EM <> FL PP ED

Label

Switch that controls the PGR file write operations :

Off
: Do not write the PGR file during the solution.

On
: Write the PGR file during solution (default).

STAT
: Display the current status of the PGR file settings.

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to *Jobname*, or *File* if no jobname is specified.

Fext

Filename extension (8 character maximum).

--

Unused field

DataType

Data type to create. This specification applies to discontinuous results data such as stress, strain, and field data. .

0
: Save as nodal-averaged data (default). Used by the **PLNSOL** command.

1
: Reserved.

2
: Save both nodal averaged and unaveraged data. Used by the **PLESOL** and **PLNSOL** commands.

InteriorKey

Key that controls whether or not internal model data is saved to the PGR file. Internal data is required in order to use graphical slicing, capping, vector displays, or Isosurface displays for data display. Interior data is also required for the **AVRES**,,FULL option, where surface results take into consideration the interior element contributions.

0
: Save exterior surface data only. (default)

1
Save exterior and interior data.

Append
Append or overwrite data to the specified file. .

0
Append the data to the existing file (default). Note that the geometry must not have been changed between the write operations.

1
Overwrite the data file.

Notes

In interactive mode, you cannot read the PGR file while **PGWRITE** is active. If you solve your model and want to plot the model while still in SOLUTION, you must first close the PGR file by issuing **PGWRITE**,Off. This occurs automatically when you exit SOLUTION and enter POST1.

When you wish to view your stress results in another coordinate system, you must generate your PGR file from the results file, in POST1, in that coordinate system.

Menu Paths

Main Menu>Solution>Output Ctrls>PGR File
Main Menu>General Postproc>Write PGR File

PHYSICS, *Option*, *Title*, *Fname*, *Ext*, --
Writes, reads, or lists all element information

PREP7: Special Purpose

MP ME ST DY <> PR EM <> FL PP ED

Option
Specifies what to do with element information:

WRITE
Write all appropriate element types, key options, real constants, material properties, solution analysis options, load step options, constraint equations, coupled nodes, defined components, and GUI preference settings to the file specified with the *Fname* and *Ext* arguments.

READ
Deletes all solution information (material properties, solution options, load step options, constraint equations, coupled nodes, results, and GUI preference settings) then reads all the information listed above into the ANSYS database from the location specified by the *Fname* and *Ext* arguments.

LIST
Lists currently defined physics files and their titles.

DELETE
Deletes a specified physics file and its title from the database.

CLEAR
: Deletes all material properties, solution options, load step options, constraint equations, coupled nodes, results, and GUI preference settings from the database. Does NOT clear the active physics file title from the database.

STATUS
: Displays information about all active elements and settings.

Title
: A user-defined title that quickly identifies a set of physics settings. For example, you might use "Fluid," "Structural," or "Magnetic" as titles. A title can contain up to 64 characters. It can be entered in lower or upper case. Lower case is internally converted to upper case within the program.

Fname
: File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

  The file name defaults to **Jobname**. Previous data on this file, if any, are overwritten.

Ext
: Filename extension (8 character maximum).

  The extension defaults to PH*n* if *Fname* is blank, where *n* is a number between one and nine, depending on how many times you have issued the **PHYSICS** command. (You can have up to nine separate physics files.) If you issue the command more than nine times, the ANSYS program will require you to delete an existing file.

--
: Unused field

Notes

Use the **PHYSICS** command when you are performing a multiphysics analysis that involves two different disciplines (for example, CFD and structural analysis) and you cannot solve both analyses simultaneously. Once you have set up physics environments for both analyses, you can use the **PHYSICS**,READ command to change between the defined physics environments. For more information about doing multiphysics analyses, see Sequential Coupled-Field Analysis in the *ANSYS Coupled-Field Analysis Guide*.

The **PHYSICS** command outputs all solution information, including analysis options, to the **Jobname.PH*n*** file described above. Although it also outputs components, the ANSYS program does not list entities (nodes, elements, lines, etc.).

PHYSICS,WRITE will overwrite existing physics files with the same title (even if the name is different). In other words, if the directory has a physics file with the same title as the active physics file title, but a different name, the **PHYSICS**,WRITE command will overwrite the existing physics file and use the existing filename, not the file-name specified on the **PHYSICS**,WRITE command.

Menu Paths

Main Menu>Preprocessor>Physics>Environment>Clear
Main Menu>Preprocessor>Physics>Environment>Delete
Main Menu>Preprocessor>Physics>Environment>List
Main Menu>Preprocessor>Physics>Environment>Read
Main Menu>Preprocessor>Physics>Environment>Status

Main Menu>Preprocessor>Physics>Environment>Write
Main Menu>Solution>Physics>Environment>Clear
Main Menu>Solution>Physics>Environment>Delete
Main Menu>Solution>Physics>Environment>List
Main Menu>Solution>Physics>Environment>Read
Main Menu>Solution>Physics>Environment>Status
Main Menu>Solution>Physics>Environment>Write

/PICE, *Item*, *--*, *KEY*

Shows initial conditions on elements as contours on displays.

GRAPHICS: Labeling

MP <> <> <> <> <> <> <> FL PP ED

Item
Label identifying the item:

VFRC
Volume fraction.

--
Unused field.

KEY
Symbol key:

0
Do not show initial condition contours.

1
Show initial condition contours.

Command Default

No initial condition contours displayed.

Notes

Shows initial conditions as contours on displays for the selected elements. Use **/PSTATUS** or **/PICE**,STAT to display settings. Use **/PICE**,DEFA to reset all specifications back to default.

/PICE is overridden if **/PBF**, **/PSF**, and **/PBC** are all on.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Symbols

PINCLUDE, *ELEM*

Specifies elements to be included in p-level escalations.

SOLUTION: p-Method

MP ME ST <> <> PR <> <> <> PP ED

ELEM
Element number of element to be included. If *ELEM* = ALL, include all elements. If *ELEM* = STAT, give status of included elements. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

Command Default

All elements are included.

Notes

The elements chosen by this command may have their p-levels increased throughout the solution iterations.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>p-Method>Stored Energy>Include Elems
Main Menu>Preprocessor>Loads>Load Step Opts>p-Method>Strain Energy>Include Elems
Main Menu>Solution>Load Step Opts>p-Method>Stored Energy>Include Elems
Main Menu>Solution>Load Step Opts>p-Method>Strain Energy>Include Elems

---

PINSUL, *DENS*, *ITK*

Defines the external insulation constants in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

DENS
Insulation density.

ITK
Insulation thickness.

Command Default

No insulation.

Notes

Defines the external insulation constants in a piping run. See the **RUN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Specifications

---

PIPE

Specifies "Pipe modeling" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Piping Module

PIVCHECK, *KEY*

Prevents a batch mode, linear static analysis from stopping when a negative or zero equation solver pivot value is encountered.

SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> <> PP ED

KEY

Determines whether to stop or continue an analysis when a negative or zero equation solver pivot value is encountered:

ON

Default value. ANSYS checks for negative or zero pivot values on linear static analyses performed with the frontal, sparse and PCG solvers. When one is encountered, an error is issued, stopping the job. A negative pivot value may be valid for some Multiphysics analyses (e.g. Emag or Thermal); this key has no effect in these cases.

OFF

Pivots will not be checked. This key continues the analysis in spite of a zero or negative pivot value. The program will proceed until other error checking routines are encountered.

Command Default

ON, stop the analysis and report an error status to the user.

Notes

This command is valid only for linear static analyses. In a nonlinear analysis, a negative pivot may be valid. Normally, rigid body motions in a nonlinear analysis will be trapped by error routines checking infinitely large displacements (DOF limit exceeded) or nonconvergence status. An under-constrained model may avoid the pivot check, but fail with a DOF limit exceeded error.

This command is applicable only to batch mode (not in the GUI). While in an ANSYS interactive session, it is generally desirable to remain in the session, even though a pivot error has occurred. Warning messages will be displayed on the screen, allowing the user to take corrective action.

Finally, machine precision will determine if a warning or an error is issued for the small pivot case. In all cases there will be some notification that the model needs to be checked closely for an accurate solution.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

PLCONV, *Item*, *Comp*, *NODE*, *Surf*

Plots the convergence curve for specified items from a p-method solution.

POST1: Results

MP ME ST DY <> PR <> <> <> PP ED

Item
Label identifying the convergence criterion to be plotted. (Valid component labels are described in PLCONV - Valid Item and Component Labels.)

ALL
Simultaneously plot the convergence curve for all specified *Item* criteria versus p-level (default). Does not require a component label.

SE
Plot strain energy curve or stored electrostatic energy curve, versus p-level. Does not require a component label.

S
Plot stress curve versus p-level. Valid component labels are described in PLCONV - Valid Item and Component Labels below.

EPEL
Plot strain curve versus p-level. Valid component labels are described in PLCONV - Valid Item and Component Labels below.

U
Plot translational displacement curve versus p-level. Valid component labels are described in PLCONV - Valid Item and Component Labels below.

ROT
Plot structural rotation versus p-level (valid only for SHELL150).

VOLT
Plot electric potential curve versus p-level. Does not require a component label.

EF
Plot electric field strength curve versus p-level. Valid component labels are described in PLCONV - Valid Item and Component Labels below.

D
Plot electric flux density curve versus p-level. Valid component labels are described in PLCONV - Valid Item and Component Labels below.

EFORC
Plot global electrostatic (Maxwell Stress Tensor) force curve versus p-level. Valid component labels are described in PLCONV - Valid Item and Component Labels below.

Comp
Component of the *Item*. Not required for *Item* = ALL, SE, VOLT, or EFORC. Valid *Comp* values (and associated *Item* values) are in PLCONV - Valid Item and Component Labels below. (Defaults to ALL: plot the convergence curve for all *Item* and *Comp* criteria simultaneously.)

NODE
Node number where convergence checking was performed. Valid only for *Item* = S, EPEL, U, ROT, VOLT, EF, or D. Defaults to all nodes having *Item* and *Comp* previously specified as convergence criteria [**PCONV**].

Surf
Surface of a p-element shell structure where the convergence curve for *NODE* will be plotted. Valid only for *Item* = S or EPEL. Defaults to all surfaces.

TOP
Top surface (default).

MID
Middle surface

BOT
Bottom surface.

Command Default

Display all specified criteria simultaneously.

Notes

The previously specified convergence criterion [**PCONV**] is graphed versus the characteristic p-level. The characteristic p-level is the polynomial level range from the minimum defined [**PPRANGE**] to the maximum p-level reached by any element in the model. The *Item* value corresponds to the convergence criterion indicated by **PCONV**. All *Item* values previously designated by the **PCONV** command will be plotted.

PLCONV - Valid Item and Component Labels

Item	Comp	Description
ALL	ALL	All convergence criteria (default)
SE		Strain energy or stored electrostatic energy
S	X, Y, Z, XY, YZ, XZ	Component stress
"	1, 2, 3	Principal stress
"	INT	Stress intensity
"	EQV	Equivalent stress
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain
"	1, 2, 3	Principal elastic strain

Item	Comp	Description
"	INT	Elastic strain intensity
"	EQV	Elastic equivalent strain
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum
"	ALL	All applicable displacement items
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum (valid only for SHELL150)
VOLT		Electric potential
EF	X, Y, Z, SUM	Electric field strength
D	X, Y, Z, SUM	Electric flux density
EFORC	X, Y, Z, SUM	Global electrostatic (Maxwell Stress Tensor) force

Menu Paths

Main Menu>General Postproc>Plot Results>p-Method>p-Convergence

---

PLCPLX, *KEY*

Specifies the part of a complex variable to display.

POST26: Display

MP ME ST DY <> PR EM <> <> PP ED

KEY

Complex variable part:

0
: Amplitude.

1
: Phase angle.

2
: Real part.

3
: Imaginary part.

Notes

Used only with harmonic analyses (**ANTYPE**,HARMIC).

All results data are stored in the form of real and imaginary components and converted to amplitude and/or phase angle as specified via the **PLCPLX** command. The conversion is not valid for derived results (such as principal stress/strain, equivalent stress/strain and USUM).

Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph

PLCRACK, *LOC*, *NUM*

Displays cracking and crushing locations in SOLID65 elements.

POST1: Special Purpose

MP ME ST <> <> <> <> <> <> PP ED

LOC

Location at which symbols are to be displayed:

0
Plot symbols at integration points (default).

1
Plot symbol at element centroids (averaged).

NUM

Crack to be displayed:

0
Plot all cracks (default).

1
Plot only the first crack.

2
Plot only the second crack.

3
Plot only the third crack.

Notes

PLCRACK displays circles at locations of cracking or crushing in concrete elements. Cracking is shown with a circle outline in the plane of the crack, and crushing is shown with an octahedron outline. If the crack has opened and then closed, the circle outline will have an X through it. Each integration point can crack in up to three different planes. The first crack at an integration point is shown with a red circle outline, the second crack with a green outline, and the third crack with a blue outline.

Symbols shown at the element centroid (*LOC* = 1) are based on the status of all of the element's integration points. If any integration point in the element has crushed, the crushed (octahedron) symbol is shown at the centroid. If any integration point has cracked or cracked and closed, the cracked symbol is shown at the element centroid. If at least five integration points have cracked and closed, the cracked and closed symbol is shown at the element centroid. Finally, if more than one integration point has cracked, the circle outline at the element centroid shows the average orientation of all cracked planes for that element.

Portions of this command are not supported by PowerGraphics [**/GRAPHICS**,POWER].

Menu Paths

Main Menu>General Postproc>Plot Results>ConcPlot>Crack/Crush

PLDISP, *KUND*

Displays the displaced structure.

POST1: Results

MP ME ST DY <> PR <> <> <> PP ED

KUND
Undisplaced shape key:

0
Display only displaced structure.

1
Overlay displaced display with similar undisplaced display (appearance is system-dependent).

2
Same as 1 except overlay with undisplaced edge display (appearance is system-dependent).

Notes

Displays the displaced structure for the selected elements.

For information on true scale plots, refer to the description of the **/DSCALE** command [**/DSCALE**,,1.0].

Menu Paths

Main Menu>General Postproc>Plot Results>Deformed Shape
Utility Menu>Plot>Results>Deformed Shape
Utility Menu>PlotCtrls>Animate>Deformed Shape

PLESOL, *Item*, *Comp*, *KUND*, *Fact*

Displays the solution results as discontinuous element contours.

POST1: Results

MP ME ST DY <> PR EM <> FL PP ED

Item
Label identifying the item. Valid item labels are shown in PLESOL - Valid Item and Component Labels below. Some items also require a component label.

Comp
Component of the item (if required). Valid component labels are shown in PLESOL - Valid Item and Component Labels below.

KUND
Undisplaced shape key:

0
Do not overlay undeformed structure display

1
Overlay displaced contour plot with undeformed display (appearance is system-dependent)

2
Overlay displaced contour plot with undeformed edge display (appearance is system-dependent)

Fact
Scale factor for 2-D display of contact items (defaults to 1). A negative scaling factor may be used to invert the display.

Notes

Displays the solution results as element contours discontinuous across element boundaries for the selected elements. For example, **PLESOL**,S,X displays the X component of stress S (i.e., the SX stress component). Various element results depend on the calculation method and the selected results location (**AVPRIN**, **RSYS**, and **ESEL**). Contours are determined by linear interpolation within each element, unaffected by the surrounding elements (i.e., no nodal averaging is performed). The discontinuity between contours of adjacent elements is an indication of the gradient across elements. Component results are displayed in the active results coordinate system [**RSYS**] (default is the global Cartesian). See the **ETABLE** and **PLETAB** commands for displaying items not available through this command (such as line element results).

For PowerGraphics displays [**/GRAPHICS**,POWER], results are plotted only for the model exterior surface. The items marked with [1] in PLESOL - Valid Item and Component Labels are not supported by PowerGraphics.

PLESOL - Valid Item and Component Labels

Item	Comp	Description
	Valid item and component labels for element results are:	
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
"	MAXF	Maximum Stress Failure Criteria. (Only works if FC command information is provided.) [1].
"	TWSI	Tsai-Wu Strength Index Failure Criterion. (Only works if FC command information is provided.) [1].
"	TWSR	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion. (Only works if FC command information is provided.) [1].
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
"	MAXF	Maximum Strain Failure Criteria. (Only works if FC command information is provided.) [1]
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT	Creep strain intensity.

Item	Comp	Description
"	EQV	Creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
EPSW		Swelling strain.
EPTO	X, Y, Z, XY, YZ, XZ	Component total mechanical strain (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total mechanical strain.
"	INT	Total mechanical strain intensity.
"	EQV	Total mechanical equivalent strain.
EPTT	X, Y, Z, XY, YZ, XZ	Total mechanical *and* thermal strain (EPEL + EPPL + EPCR + EPTH).
"	1, 2, 3	Principal total mechanical *and* thermal strain.
"	INT	Total mechanical *and* thermal strain intensity.
"	EQV	Total mechanical *and* thermal equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
SVAR	1, 2, 3, ... N	State variable.
GKS	X, XY, XZ	Gasket component stress.
GKD	X, XY, XZ	Gasket component total closure.
GKDI	X, XY, XZ	Gasket component total inelastic closure.
GKTH	X, XY, XZ	Gasket component thermal closure.
CONT	STAT	Contact status. 3 - closed and sticking, 2 - closed and sliding, 1 - open but near contact, 0 - open and not near contact.
"	PENE	Contact penetration
"	PRES	Contact pressure
"	SFRIC	Contact friction stress
"	STOT	Contact total stress (pressure plus friction)
"	SLIDE	Contact sliding distance
"	GAP	Contact gap distance
"	FLUX	Total heat flux at contact surface
"	CNOS	Total number of contact status changes during substep.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.

Item	Comp	Description
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component magnetic force or vector sum [1].
P	X, Y, Z, SUM	Pointing vector component or sum [1].
SERR		Structural error energy [1].
SDSG		Absolute value of the maximum variation of any nodal stress component [1].
TERR		Thermal error energy [1].
TDSG		Absolute value of the maximum variation of any nodal thermal gradient component [1].
F	X, Y, Z	X, Y, or Z structural force [1].
M	X, Y, Z	X, Y, or Z structural moment [1].
HEAT		Heat flow [1].
FLOW		Fluid flow [1].
AMPS		Current flow [1]. Use **FORCE** for type.
CHRG		Charge [1]. Use **FORCE** for type.
FLUX		Magnetic flux [1].
VF	X, Y, Z	X, Y, or Z fluid "force" component [1].
CSG	X, Y, Z	X, Y, or Z magnetic current segment component [1].
SENE		"Stiffness" energy or thermal heat dissipation. Same as TENE[1].
TENE		Thermal heat dissipation or "stiffness" energy. Same as SENE [1].
KENE		Kinetic energy [1].
JHEAT		Element Joule heat generation [1].
JS	X, Y, Z, SUM	Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low frequency electric analyses. Components (X, Y, Z) and vector sum (SUM). [1].
JT	X, Y, Z, SUM	Total measureable current density in low-frequency electromagnetic analyses. (Conduction current density in a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM). [1].
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM). [1].
MRE		Magnetic Reynolds number [1].
VOLU		Volume of volume element [1].
CENT	X, Y, Z	Centroid X, Y, or Z location (based on shape function) in the active coordinate system [1].
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
SMISC	*snum*	Element summable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each element description found in the *ANSYS Elements Reference*).

Item	Comp	Description
NMISC	*snum*	Element non-summable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each element description found in the *ANSYS Elements Reference*).
TOPO		Densities used for topological optimization. This applies to the following types of elements: PLANE2, PLANE82, SOLID92, SHELL93, SOLID95.

1. Not supported by PowerGraphics

Menu Paths

Main Menu>General Postproc>Plot Results>Contour Plot>Element Solu
Utility Menu>Plot>Results>Contour Plot>Elem Solution

PLETAB, *Itlab*, *Avglab*

Displays element table items.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

Itlab
User-defined label, as specified with the **ETABLE** command, of item to be displayed.

Avglab
Averaging operation:

NOAV
Do not average element items at common nodes (default).

AVG
Average the element items at common nodes.

Notes

Displays items stored in the table defined with the **ETABLE** command for the selected elements. For display purposes, items are assumed to be constant over the element and assigned to each of its nodes. Contour display lines (lines of constant value) are determined by linear interpolation within each element from the nodal values. These nodal values have the option of being averaged (values are averaged at a node whenever two or more elements connect to the same node) or not averaged (discontinuous). The discontinuity between contour lines of adjacent elements is an indication of the gradient across elements.

Portions of this command are not supported by PowerGraphics [**/GRAPHICS**,POWER].

Menu Paths

Main Menu>General Postproc>Element Table>Plot Elem Table
Main Menu>General Postproc>Plot Results>Contour Plot>Elem Table
Utility Menu>Plot>Results>Contour Plot>Elem Table Data

PLF2D, *NCONT*, *OLAY*, *ANUM*, *WIN*

Generates a contour line plot of equipotentials.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

NCONT
Number of contour lines to display. Issue in multiples of 9 (i.e., 9, 18, 27, etc.). Default is 27 contour lines.

OLAY
Overlay:

0
Overlay edge outlines by material number.

1
Overlay edge outlines by real constant number.

ANUM
Highest material or real constant attribute number. Command will cycle through *ANUM* element display overlays. Defaults to 10.

WIN
Window number to which command applies. Defaults to 1.

Notes

PLF2D invokes an ANSYS macro which plots equipotentials of the degree of freedom AZ. These equipotential lines are parallel to flux lines and thus give a good representation of flux patterns. In the axisymmetric case, the display is actually r*AZ where "r" is the node radius. The macro overlays (*OLAY*) edge outlines by material number or real constant number (*ANUM*) and allows user control over the number of contour lines to display (*NCONT*).

Menu Paths

Main Menu>General Postproc>Plot Results>Contour Plot>2D Flux Lines
Utility Menu>Plot>Results>Flux Lines

PLHFFAR, *Opt*, *Lab*, *PHI1*, *PHI2*, *NPHI*, *THETA1*, *THETA2*, *NTHETA*, *RADIUS*, *CS*, *Comp*

Displays electromagnetic far fields far field parameters.

POST1: Special Purpose

MP <> <> <> <> <> <> EH <> PP ED

Opt
Display option:

FIELD
Electromagnetic far field.

RCS
Radar cross section.

RCSN
Normalized radar cross section.

PATT
Antenna radiation pattern.

DGAIN
Antenna directive gain.

Lab
As shown below, data entered in the Lab field will vary, depending on the display option (Opt).

Valid Lab Data Labels

Opt	*Lab*
FIELD	EF -- Electric field (default) H -- Magnetic field
RCS and RCSN	NONE -- Radar echo area (default) PP -- Phi-Phi polarization PT -- Phi-Theta polarization TP -- Theta-Phi polarization TT -- Theta-Theta polarization
PATT	RECT -- Cartesian coordinate system plot (default) POLAR -- Polar coordinate system plot
DGAIN	RECT -- Cartesian coordinate system plot (default) POLAR -- Polar coordinate system plot

PHI1, *PHI2*
Starting and ending ϕ angles (degrees) in the spherical coordinate system. Defaults to 0.

NPHI
Number of divisions between the starting and ending ϕ angles for data computations. Defaults to 0.

THETA1, *THETA2*
Starting and ending θ angles (degrees) in the spherical coordinate system. Defaults to 0.

NTHETA
Number of divisions between the starting and ending θ angles for data computations. Defaults to 0.

The following arguments are used only with *Opt* = FIELD.

RADIUS
Radius for far field. Defaults to 1.

CS
Coordinate system type:

0
Cartesian (default).

2
Spherical.

Comp
Coordinate system component:

SUM
Magnitude of electromagnetic field (default).

X
X-component (CS = 0) or radial component (CS = 2).

Y
Y-component (CS = 0) or ϕ component (CS = 2).

Z
Z-component (CS = 0) or θ component (CS = 2).

See Spherical Coordinates in the *ANSYS Low-Frequency Electromagnetic Analysis Guide*.

Notes

PLHFFAR displays electromagnetic far field and far field parameters as determined by the equivalent source principle. Use this command to display far electromagnetic field, radar cross section, antenna radiation pattern, or antenna directive gain.

Menu Paths

Main Menu>General Postproc>Plot Results>Field Extension>Direct Gain
Main Menu>General Postproc>Plot Results>Field Extension>Far Field
Main Menu>General Postproc>Plot Results>Field Extension>Pattern
Main Menu>General Postproc>Plot Results>Field Extension>RCS
Main Menu>General Postproc>Plot Results>Field Extension>RCS Normalized

---

PLLS, *LabI*, *LabJ*, *Fact*, *KUND*

Displays element table items as contoured areas along elements.

POST1: Element Table

MP ME ST DY <> PR <> <> <> PP ED

LabI
Label of element table item [**ETABLE**] for node I magnitude.

LabJ
Label of element table item for node J magnitude.

Fact
Scale factor for display (defaults to 1). A negative scaling factor may be used to invert the display.

KUND
Undisplaced shape key:

0
Do not overlay undeformed structure display

1
Overlay displaced contour plot with undeformed display (appearance is system-dependent)

2
Overlay displaced contour plot with undeformed edge display (appearance is system-dependent)

Notes

Displays selected items (e.g., shears and moments) as a contoured area (trapezoid) display along line elements and 2-D axisymmetric shell elements (e.g., shear and moment diagrams). Three sides of the trapezoid are formed by the element (one side) and lines at nodes I and J of length proportional to the item magnitude and displayed normal to the element and the viewing direction (the two parallel sides).

Portions of this command are not supported by PowerGraphics [**/GRAPHICS**,POWER].

Menu Paths

Main Menu>General Postproc>Plot Results>Contour Plot>Line Elem Res

---

PLNSOL, *Item*, *Comp*, *KUND*, *Fact*, *FileID*

Displays results as continuous contours.

POST1: Results

MP ME ST DY <> PR EM <> FL PP ED

Item
Label identifying the item. Valid item labels are shown in PLNSOL - Valid Item and Component Labels below. Some items also require a component label.

Comp
Component of the item (if required). Valid component labels are shown in PLNSOL - Valid Item and Component Labels below.

KUND
Undisplaced shape key:

0
Do not overlay undeformed structure display

1
Overlay displaced contour plot with undeformed display (appearance is system-dependent)

2
Overlay displaced contour plot with undeformed edge display (appearance is system-dependent)

Fact
Scale factor for 2-D display for contact items. Default value is 1. A negative scaling factor may be used to invert the display.

FileID
The file index number (obtained via the **NLDIAG**,NRRE,ON command). Valid only for *Item* = NRRE.

Notes

Displays the solution results as continuous contours across element boundaries for the selected nodes and elements. For example, **PLNSOL**,S,X displays the X component of stress S (that is, the SX stress component). Various

element results depend upon the recalculation method and the selected results location [**AVPRIN**, **RSYS**, **LAYER**, **SHELL**, and **NSEL**]. Contours are determined by linear interpolation within each element from the nodal values, which are averaged at a node whenever two or more elements connect to the same node (except for FMAG, which is summed at the node).

For PowerGraphics displays [**/GRAPHICS**,POWER], results are plotted only for the model exterior surface. The items marked with [2] are not supported by PowerGraphics. To plot midside nodes, you must first issue **/EFACET**,2.

PLNSOL - Valid Item and Component Labels

Item	Comp	Description
	Valid item and component labels for nodal degree of freedom results are:	
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum.
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum.
TEMP[1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum in a fluid analysis, or X, Y, or Z nodal velocity or vector sum in a structural transient dynamic analysis (LS-DYNA analysis or ANSYS analysis with **ANTYPE**,TRANS).
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum in an electromagnetic analysis, or X, Y, or Z acceleration or vector sum in a structural transient dynamic analysis (LS-DYNA analysis or ANSYS analysis with **ANTYPE**,TRANS).
ENKE		Turbulent kinetic energy (FLOTRAN).
ENDS		Turbulent energy dissipation (FLOTRAN).
SP0*n*		Mass fraction of species *n*, where *n* = 1 to 6 (FLOTRAN). If a species is given a user-defined name [**MSSPEC**], use that name instead of *n*.
WARP		Warping.
NRRE	FX, FY, FZ, FNRM, MX, MY, MZ, MNRM	Plot the Newton-Raphson residuals from the file you obtained via the **NLDIAG**,NRRE,ON command. The FNRM and MNRM labels are computed as the square root of the sum of the squares of the residual component forces or moments (FX,FY,FZ, MX, MY, MZ). [4] When KUND = 0, use the absolute value of the residual from the files (default).
	Valid item and component labels for element results are:	
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT	Stress intensity.
"	EQV	Equivalent stress.
"	MAXF	Maximum Stress Failure Criteria. (Only works if FC command information is provided.) [2].
"	TWSI	Tsai-Wu Strength Index Failure Criterion. (Only works if FC command information is provided.) [2].
"	TWSR	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion. (Only works if FC command information is provided.) [2].

Item	Comp	Description
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
"	MAXF	Maximum Strain Failure Criteria. (Only works if FC command information is provided.) [2]
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
EPSW		Swelling strain.
EPTO	X, Y, Z, XY, YZ, XZ	Component total mechanical strain (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total mechanical strain.
"	INT	Total mechanical strain intensity.
"	EQV	Total mechanical equivalent strain.
EPTT	X, Y, Z, XY, YZ, XZ	Component total mechanical *and* thermal strain (EPEL + EPPL + EPCR + EPTH).
"	1, 2, 3	Principal total mechanical *and* thermal strain.
"	INT	Total mechanical *and* thermal strain intensity.
"	EQV	Total mechanical *and* thermal equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
SVAR	1, 2, 3, ... N	State variable.
GKS	X, XY, XZ	Gasket component stress.
GKD	X, XY, XZ	Gasket component total closure.
GKDI	X, XY, XZ	Gasket component total inelastic closure.

Item	Comp	Description
GKTH	X, XY, XZ	Gasket component thermal closure.
For contact results PowerGraphics is applicable for 3-D models only.		
CONT	STAT[3]	Contact status. 3-closed and sticking, 2-closed and sliding, 1-open but near contact, 0-open and not near contact.
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
"	GAP	Contact gap distance.
"	FLUX	Total heat flux at contact surface.
"	CNOS	Total number of contact status changes during substep.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component magnetic force or vector sum [2].
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM). [2].
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
TOPO		Densities used for topological optimization. This applies to nodes attached to the following types of elements: PLANE2, PLANE82, SOLID92, SHELL93, SOLID95.
	Valid item labels for FLOTRAN nodal results are:	
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
SPHT		Specific heat [2].
EVIS		Fluid effective viscosity.
CMUV		Turbulent viscosity coefficient.
ECON		Fluid effective conductivity.

Item	Comp	Description
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.
SFTS		Surface tension coefficient.
LMD*n*		Laminar mass diffusion coefficient for species *n*, where *n* = 1 to 6.
EMD*n*		Effective mass diffusion coefficient for species *n*, where *n* = 1 to 6.
RDFL		Radiation heat flux [2].

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . ., TTOP instead of TEMP to view the individual temperature degree of freedom. When other thermal elements are included in the model, they should be unselected to avoid plotting undefined information. To view all temperatures in the same plot, set **/ESHAPE**,1 and **/GRAPHICS**,POWER and issue **PLNSOL**,TEMP.
2. Not supported by PowerGraphics
3. For the CONT items for elements CONTA171 through CONTA175, the reported data is averaged across the element.
4. When plotting Newton-Raphson residual items (*Item* = NRRE) from a file on the deformed geometry, the displacements are based on the current set of results in the database. These displacements may not correspond to the loadstep and substep in the **.nr*xxxxx*** file. For more information about **.nr*xxxxx*** files and nonlinear diagnostics postprocessing, see the description of the **NLDPOST** command and Section 8.10.2.1: Performing Nonlinear Diagnostics.

Menu Paths

Main Menu>Drop Test>Animate Results
Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu
Utility Menu>Plot>Results>Contour Plot>Nodal Solution
Utility Menu>PlotCtrls>Animate>Animate Over Results
Utility Menu>PlotCtrls>Animate>Animate Over Time
Utility Menu>PlotCtrls>Animate>Deformed Results
Utility Menu>PlotCtrls>Animate>Dynamic Results
Utility Menu>PlotCtrls>Animate>Isosurfaces
Utility Menu>PlotCtrls>Animate>Mode Shape
Utility Menu>PlotCtrls>Animate>Q-Slice Contours
Utility Menu>PlotCtrls>Animate>Time-harmonic

/PLOPTS, *Label*, *KEY*

Controls graphics options on subsequent displays.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

Label

Apply display items as selected from the following labels:

INFO

Controls the display of the legend (ON or OFF) and allows the choice of preset or Multi-legend placement. Control is provided by the *KEY* values. (Defaults to *KEY*=3 when the GUI is on. Defaults to *KEY*= 2 otherwise.)

LEG1
Header portion of legend column (defaults to ON).

LEG2
View portion of legend column (defaults to ON (except off with contour displays)).

LEG3
View the contour section of the legend column (defaults to ON).

FRAME
Frame border lines around windows (defaults to ON).

TITLE
Title (bottom left text) (defaults to ON).

MINM
Min-Max symbols on contour displays (defaults to ON).

LOGO
ANSYS logo (defaults to OFF (displayed as text at top of legend column)). If *KEY* = ON, the text is removed from legend column but the logo symbol is displayed in whichever active window is either in the upper-most right corner and on top, or if there is no window in that location, then in the window to the furthest right of the screen. Version information remains in the legend column.

WINS
Controls whether graphics windows automatically stretch or shrink to adjust to screen size as the legend column is turned off or on [**/PLOPTS**,INFO] (defaults to ON). If WINS is on and the legend column is changed from off to on, all windows are shrunk regardless of what their correct size is.

WP
Working plane (defaults to OFF). The working plane is drawn as part of the display (*not* just an overlaid image as in **WPSTYL**). This option is best used in combination with a hidden-line technique [**/TYPE**].

DATE
Controls the display of the date and time in your legend. Subsequent *KEY* values control the display as follows:

Off or 0 - No date or time displays are included in your legend.
1 - Only the date is shown.
2 (default) - both the date and time are shown.

FILE
Controls the display of the ANSYS jobname in your legend. Subsequent *KEY* values control the display as follows:

Off or 0 (default) - The ANSYS jobname is NOT included in your legend.
On or 1 - The ANSYS jobname is included in your legend.

KEY
Switch:

OFF or 0
Do not apply this display item. For *Label* = DATE, no time or date are displayed.

ON or 1
Apply this display item. For *Label* = DATE, show only the date.

AUTO or 2
For *Label* = INFO, initiate Auto-legend mode. If the display has contours, the legend is ON; if the display has no contours, the legend is OFF. For *Label* = DATE, display both the date and time.

3
For *Label* = INFO , switch to Multi-legend mode. See the **/UDOC** command for the available legend configurations.

Command Default

See individual label defaults.

The Multi-legend mode (**/PLOPTS**,INFO,3) is the default for contour legend displays.

Notes

Use **/PLOPTS**,STAT to display settings. Use **/PLOPTS**,DEFA to reset all specifications back to their defaults.

When you perform multiple results displays, contours on the legend column may be truncated. To avoid this, specify **/PLOPTS**,LEG1,0.

The Multi-legend mode provides a number of legend data item priority and placement options. These options are accessed via the GUI at **Utility Menu> PlotCtrls> Window Controls> Window Options**. The **/UDOC** command provides command line options for this capability.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Window Controls>Reset Window Options
Utility Menu>PlotCtrls>Window Controls>Window Options

PLOT, *NSTRT*, *NEND*, *NINC*

Forms a display.

DISPLAY: Action

MP ME ST DY <> PR EM <> FL PP ED

NSTRT, *NEND*, *NINC*
Display plots sequentially from number *NSTRT* to *NEND* in steps of *NINC*. *NSTRT* defaults to the next plot. *NEND* defaults to *NSTRT*. *NINC* defaults to 1. If *NSTRT* = ALL, display all plots from the beginning of the file. If *NEND* = ALL, display to the end of the file.

Notes

Output will be to the terminal or to a file, depending on the driver [**/SHOWDISP**]. The INTERLEAF and DUMP drivers produce an output file for each plot named INTL*nn* and DUMP*nn*, with *nn* sequentially ranging from 00 to 99. A blank line after the **PLOT** command causes the next plot to be formed.

Menu Paths

It is part of the DISPLAY command.

PLOTTING

Specifies "Plotting settings" as the subsequent status topic.

POST26: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>TimeHist Postproc>Plot

PLPAGM, *Item*, *Gscale*, *Nopt*

Displays path items along the path geometry.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

Item

The path data item to be displayed on the currently active path (defined by the **PATH** command). Valid path items are those defined with the **PDEF** or **HFNEAR** commands.

Gscale

Scale factor for the offset from the path for the path data item displays. Defaults to 1.0.

Nopt

Determines how data is displayed:

(blank)

Do not display nodes, and scale the display based on the currently selected node set (default).

NODE

Display path item data along with the currently selected set of nodes. The display geometry is scaled to the selected node set.

Notes

You can use the *Gscale* argument to scale the contour display offset from the path for clarity. You need to type all six characters to issue this command.

Menu Paths

Main Menu>General Postproc>Path Operations>Plot Path Item>On Geometry
Main Menu>General Postproc>Plot Results>Plot Path Item>On Geometry

---

PLPATH, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

Displays path items on a graph.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

Lab1, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*
Labels identifying the path items to be displayed. Up to six items may be drawn per frame. Predefined path geometry items XG, YG, ZG, and S [**PDEF**] may also be displayed.

Notes

The path must have been defined by the **PATH** and **PPATH** commands. Path items and their labels must have been defined with the **PDEF**, **HFNEAR**, **PVECT**, **PCALC**, **PDOT**, and **PCROSS** commands. Path items may also be printed with the **PRPATH** command. Graph scaling may be controlled with the **/XRANGE**, **/YRANGE**, and **PRANGE** commands. You need to type all six characters to issue this command.

Menu Paths

Main Menu>General Postproc>Path Operations>Plot Path Item>On Graph
Main Menu>General Postproc>Plot Results>Plot Path Item>On Graph
Utility Menu>Plot>Results>Path Plot

PLSECT, *Item*, *Comp*, *RHO*, *KBR*

Displays membrane and membrane-plus-bending linearized stresses.

POST1: Path Operations

MP ME ST DY <> PR <> <> <> PP ED

Item
Label identifying the item to be processed. Valid item labels are shown in PLSECT - Valid Item and Component Labels below. Items also require a component label.

Comp
Component of the item. Valid component labels are shown in PLSECT - Valid Item and Component Labels below.

RHO
In-plane (X-Y) average radius of curvature of the inside and outside surfaces of an axisymmetric section. If zero (or blank), a plane or 3-D structure is assumed. If nonzero, an axisymmetric structure is assumed. Use a very large number (or -1) for an axisymmetric straight section.

KBR
Through-thickness bending stresses key for an axisymmetric analysis (*RHO* $\neq$0):

0
Include the thickness-direction bending stresses.

1
Ignore the thickness-direction bending stresses.

Notes

Calculates and displays the membrane and membrane-plus-bending linearized stresses (as described for the **PRSECT** command) along a path section [**PATH**] as a graph. The path section is defined by two points specified with the **PPATH** command. For linearized stress calculations, the path must be defined with nodes. The path must be entirely within the selected elements (that is, there must not be any element gaps along the path). The total stress (equivalent to the **PLPATH** display) is also displayed. This command always uses 48 divisions along the path, regardless of the number of divisions defined by **PATH**.

Portions of this command are not supported by PowerGraphics [**/GRAPHICS**,POWER].

PLSECT - Valid Item and Component Labels

Item	Comp	Description
	Valid item and component labels for element results are:	
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.

Menu Paths

Main Menu>General Postproc>Path Operations>Linearized Strs
Main Menu>General Postproc>Plot Results>Plot Path Item>Lineariz Strs

---

PLSP, *Opt*, *Pnum1*, *Pnum2*, *Pnum3*, *Pnum4*, *Pnum5*, *Pnum6*
Displays S-parameters on an XY graph.

POST1: Results

MP <> <> <> <> <> <> EH <> PP ED

Opt

0
Display the S-parameter magnitude.

1
Display the S-parameter magnitude in dB format.

2
Display the S-parameter phase angle (degrees).

Pnum1, *Pnum2*, *Pnum3*, *Pnum4*, *Pnum5*, *Pnum6*

Pnum1
Driven port number associated with the S-parameter.

Pnum2, *Pnum3*, *Pnum4*, *Pnum5*, *Pnum6*
Matched port number associated with the S-parameter. Up to five S-parameters may be plotted at one time. For example, if the driven port is port number 1, and *Pnum2* = 3, then S31 is displayed. If *Pnum2* = 2, and *Pnum3* = 3, then S21 and S31 are displayed.

Notes

S-parameter data is read from an S-parameter file with a filename **Jobname.sprm**. The S-parameter file is created using the **SPSWP** command.

Menu Paths

Main Menu>General Postproc>Plot Results>S-Parameters

PLTIME, *TMIN*, *TMAX*

Defines the time range for which data are to be displayed.

POST26: Display

MP ME ST DY <> PR EM <> FL PP ED

TMIN
Minimum time (defaults to the first point stored).

TMAX
Maximum time (defaults to the last point stored).

Command Default

Use the previously defined range [**TIMERANGE**].

Notes

Defines the time (or frequency) range (within the range stored) for which data are to be displayed. Time is always displayed in the Z-axis direction for 3-D graph displays. If XVAR = 1, time is also displayed in the X-axis direction and this control also sets the abscissa scale range.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph

PLTRAC, *Analopt*, *Item*, *Comp*, *TRPNum*, *Name*, *MXLOOP*, *TOLER*, *OPTION*, *ESCL*, *MSCL*

Displays a particle flow or charged particle trace on an element display.

POST1: Trace Points

MP ME ST DY <> <> EM <> FL PP ED

Analopt
Analysis option

FLUID
Particle trace in fluid flow (default)

ELEC
Particle trace in electric field

MAGN
Particle trace in magnetic field

EMAG
Particle trace in presence of both electric and magnetic fields

Item
Label identifying the item to be contoured. Valid item labels are shown in PLTRAC - Valid Item and Component Labels below. Some items also require a component label. If *Item* is blank, display only the path trajectory.

Comp
Component of the item (if required). Valid component labels are shown in PLTRAC - Valid Item and Component Labels below.

TRPNum
Trace point number for storing trajectory data for use with **PATH** logic. Defaults to 0 (no trajectory path data is stored for further processing with **PATH** logic).

Name
Name of prefix of array variable. Defaults to TRAC. *Name*POIN stores trajectory path points for trace point number *TRPNum*. If *Analopt* = ELEC, MAGN, or EMAG, two additional array parameters, *Name*DATA and *Name*LABL, store trajectory path data and labels for the same *TRPNum*.

MXLOOP
Maximum number of loops traced by a particle. Defaults to 25 for *Opt* = FLUID; otherwise, defaults to 1000.

TOLER
Length tolerance used for particle trajectory geometry calculation. Valid only for *Analopt* = ELEC, MAGN, or EMAG. If particle trace appears to terminate inside an element, adjusting the length tolerance may be necessary. Defaults to 1.0×10^{-8}.

OPTION
Flow trace option:

0
Use the undeformed mesh for computing the flow trace.

1
Use the deformed mesh for computing the flow trace.

ESCL
Electric field scale factor. Setting this scale factor affects only the tracing, not the field solution results. A negative factor corresponds to the opposite vector direction. Valid only for *Analopt* = ELEC or EMAG. Defaults to 1.

MSCL
Magnetic field scale factor. Setting this scale factor affects only the tracing, not the field solution results. A negative factor corresponds to the opposite vector direction. Valid only for *Analopt* = MAGN or EMAG. Defaults to 1.

Notes

For a specified item, the variation of the item is displayed along the particle trace as a color-contoured ribbon. The **TRPOIN** command must be used to define a point on the trajectory path. Multiple traces may be displayed simultaneously by defining multiple trace points. Issue the **TRPLIS** command to list the current tracing points. Issue the **TRPDEL** command to delete tracing points defined earlier. Use the **PAPUT** command with the POIN option to retrieve the particle trajectory points as path points.

Three array parameters are created at the time of the particle trace: TRACPOIN, TRACDATA and TRACLABL. These array parameters can be used to put the particle velocity and the elapsed time into path form. The procedure to put the arrays into a path named PATHNAME is as follows:

```
*get,npts,PARM,TRACPOIN,DIM,x
PATH,PATHNAME,npts,9,1
PAPUT,TRACPOIN,POINTS
PAPUT,TRACDATA,TABLES
PAPUT,TRACLABL,LABELS
PRPATH,S,T_TRACE,VX_TRACE,VY_TRACE,VZ_TRACE,VS_TRACE
```

Not used if *Analopt* = FLUID. If working in the GUI, use the "All information" option to retrieve information from all three arrays at once.

If *OPTION* is set to 1, the deformed mesh is based on the displacement degrees of freedom UX, UY, and UZ, which must be available in the load step.

PLTRAC - Valid Item and Component Labels

Item	Comp	Description
		Valid item and component labels for nodal degree of freedom results are:
TEMP		Temperature.
PRES		Pressure.
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum.
ENKE		Turbulent kinetic energy.
ENDS		Turbulent energy dissipation.
		Valid item and component labels for FLOTRAN nodal results are:
TTOT		Total temperature.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
SPHT		Specific heat.
EVIS		Fluid effective viscosity.
CMUV		Turbulent viscosity coefficient.
ECON		Fluid effective conductivity.
		Valid item labels for *Analopt* = ELEC nodal results are:
VOLT		Electric potential.

Item	Comp	Description
		Valid item labels for *Analopt* = MAGN or EMAG nodal results are:
None		Color contour displayed.

See the *ANSYS Basic Analysis Guide* for more information on particle flow and charged particle traces. See Animation in the *ANSYS Basic Analysis Guide* for information on particle trace animation.

Menu Paths

Main Menu>General Postproc>Plot Results>Particle Trace
Main Menu>General Postproc>Plot Results>Plot Flow Tra
Utility Menu>Plot>Results>Flow Trace
Utility Menu>PlotCtrls>Animate>Particle Flow

PLVAR, *NVAR1*, *NVAR2*, *NVAR3*, *NVAR4*, *NVAR5*, *NVAR6*, *NVAR7*, *NVAR8*, *NVAR9*, *NVAR10*
Displays up to ten variables in the form of a graph.

POST26: Display

MP ME ST DY <> PR EM <> FL PP ED

NVAR1, *NVAR2*, *NVAR3*, *NVAR4*, *NVAR5*, *NVAR6*, *NVAR7*, *NVAR8*, *NVAR9*, *NVAR10*
Variables to be displayed, defined either by the reference number or a unique thirty-two character name. If duplicate names are used the command will plot the data for the lowest-numbered variable with that name.

Notes

Variables are displayed vs. variable *N* on the **XVAR** command. The string value will be a predefined, unique name. For complex variables, the amplitude is displayed by default [**PLCPLX**]. Each **PLVAR** command produces a new frame. See the **/GRTYP** command for displaying multiple variables in a single frame with separate Y-axes.

Menu Paths

Main Menu>TimeHist Postpro>Graph Variables

PLVAROPT, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*, *Lab10*
Displays up to ten parameters in the form of a graph.

OPTIMIZATION: Display

MP ME ST DY <> PR EM <> FL PP ED

Lab1, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*, *Lab10*
Names of the parameters to be displayed on this frame.

Command Default

No display.

Notes

Displays up to ten parameters in the form of a graph. Parameters are displayed vs. *Lab* on the **XVAROPT** command (defaults to set number) in the order corresponding to an ascending order of the **XVAROPT** parameter. See the **/GRTYP** command for displaying multiple parameters in a single frame with separate Y-axes. Each **PLVAROPT** command produces a new frame.

Menu Paths

Main Menu>Design Opt>Design Sets>Graphs/Tables

PLVECT, *Item*, *Lab2*, *Lab3*, *LabP*, *Mode*, *Loc*, *Edge*

Displays results as vectors.

POST1: Results
POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

Item
Predefined vector item (from PLVECT - Valid Item Labels below) or a label identifying the i-component of a user-defined vector.

Lab2
Label identifying the j-component of a user-defined vector. Must be blank if *Item* is selected from list below.

Lab3
Label identifying the k-component of a user-defined vector. Must be blank if *Item* is selected from list below or for 2-D user defined vector.

LabP
Label assigned to resultant vector for display labeling (defaults to *Item*).

Mode
Vector or raster mode override key:

(blank)
Use the setting of *KEY* on the **/DEVICE** command.

RAST
Use raster mode for **PLVECT** displays.

VECT
Use vector mode for **PLVECT** displays.

Loc
Vector location for display of field element results:

ELEM
Display at element centroid (default).

NODE
Display at element nodes.

Edge
Edge display override key:

(blank)
Use the setting of Key on the **/EDGE** command.

OFF
Deactivate the edge display.

ON
Activate the edge display.

Notes

Displays various solution results as vectors (arrows) for the selected nodes and/or elements (elements must contain at least three nodes that are not colinear). For example, **PLVECT**,U displays the displacement vector for all selected nodes. For section displays [**/TYPE**], the vectors are shown only on the section face (i.e., cutting plane). The **PLVECT** display of principal strains and stresses (*Item* = S, EPTO, EPEL, EPPL, EPCR, or EPTH) on a "cut" of the model (**/TYPE**,,1 ,5,7,8, or 9) is not supported. The resulting plot displays the vectors on *all* selected elements, not on just the sliced surface. See the **/VSCALE** command to scale vector lengths. Vector magnitudes may be shown as a contour display with the **PLNSOL** command. Various results also depend upon the recalculation method and the selected results location [**LAYER**, **SHELL**, and **NSEL**].

Items may be selected from a set of recognized vector labels (*Item*) or a vector may be defined from up to three scalar labels (*Item*,*Lab2*,*Lab3*). Scalar labels may be user-defined with the **ETABLE** command. The vectors appear on an element display as arrows showing the relative magnitude of the vector and its direction. The predefined items will be shown either at the node or at the element centroid, depending on what item is being displayed and depending on the *Loc* setting. User defined **ETABLE** items will be shown at the element centroid, regardless of the *Loc* setting. Stress vectors appear as arrows at the element centroid, with the arrowheads pointing away from each other for tension and toward each other for compression.

For PowerGraphics, vector arrow displays are generated in Global Cartesian (**RSYS** = 0). All subsequent displays will revert to your original coordinate system.

When vector mode is active (*Mode* = VECT), use the Z-buffered display type [**/TYPE**,,6] to maximize speed of **PLVECT** plots (other hidden display types may make plotting slow). For PowerGraphics [**/GRAPHICS**,POWER], the items marked with [1] are not supported by PowerGraphics.

PLVECT - Valid Item Labels

Item	Description
	Valid item labels for nodal degree of freedom vector results are:
U	Structural displacement vector.
ROT	Structural rotation vector.
V	Velocity vector.
A	Magnetic vector potential vector.
	Valid item labels for structural element results are:
S	Principal stresses[1].
EPTO	Principal total strain (EPEL + EPPL + EPCR)[1].
EPEL	Principal elastic strains[1].
EPPL	Principal plastic strains[1].
EPCR	Principal creep strains[1].
EPTH	Principal thermal strains[1].

Item	Description
	Valid item labels for field element results are:
TG	Thermal gradient vector.
TF	Thermal flux vector.
PG	Pressure gradient vector.
EF	Electric field vector.
D	Electric flux density vector.
H	Magnetic field intensity vector. If Lab2 is blank, then Item is interpreted as one of the predefined labels. Otherwise, Item is interpreted as a user-defined **ET** label and ANSYS will request a nonblank Lab2/Lab3 according to the dimension of the problem.
B	Magnetic flux density vector.
FMAG	Magnetic force vector.
P	Poynting vector.
JS	Source current density vector for low-frequency magnetic analyses. Total current density vector (sum of conduction and displacement current densities) in low frequency electric analyses.
JT	Total measureable current density vector in low-frequency electromagnetic analyses. (Conduction current density vector in a low-frequency electric analysis.)
JC	Conduction current density vector for elements that support conduction current calculation.

1. Not supported by PowerGraphics

Menu Paths

Main Menu>General Postproc>Plot Results>Vector Plot>Predefined
Main Menu>General Postproc>Plot Results>Vector Plot>User-defined
Utility Menu>Plot>Results>Vector Plot
Utility Menu>PlotCtrls>Animate>Q-Slice Vectors

PLVFRC, *CONT*

Displays volume fractions in a volume of fluid (VOF) analysis.

PREP7: FLOTRAN Miscellaneous

MP <> <> <> <> <> <> <> FL PP ED

CONT

Contour setting:

0

Smooth plot with contour levels at 0.5 and 1.0 (default).

1

Smooth plot with current settings of contour levels.

Notes

The **PLVFRC** command macro places the volume fraction results into an element table and then plots the element table data (i.e., **ETABLE**,VFRC,NMISC,1 and then **PLETAB**,VFRC,1). If *CONT* = 0 (default), the command macro issues the /CVAL,ALL,0.5,1.0 command before the **ETABLE** command. This means **PLETAB** uses contour levels of 0.5 and 1.0 to indicate the free surface and fluid regions. If *CONT* = 1, **PLETAB** uses the current contour settings.

Menu Paths

Main Menu>Drop Test>Animate Results
Main Menu>General Postproc>Plot Results>Contour Plot>Volume Fraction
Utility Menu>PlotCtrls>Animate>Animate Over Results
Utility Menu>PlotCtrls>Animate>Animate Over Time
Utility Menu>PlotCtrls>Animate>Time-harmonic
Utility Menu>PlotCtrls>Multi-Plot Contrls

PLWAVE, *Ex*, *Ey*, *Ez*, *AngX*, *AngZ*

Specifies a free-space time-harmonic incident plane electromagnetic wave.

SOLUTION: Misc Loads

MP <> <> <> <> <> <> EH <> PP ED

Ex
Electric field amplitude in x direction.

Ey
Electric field amplitude in y direction.

Ez
Electric field amplitude in z direction.

AngX
Angle between incident wave vector and X-axis (Phi).

AngZ
Angle between incident wave vector and Z-axis (Theta).

Notes

Defines an incident plane wave for the entire solution domain. See Spherical Coordinates in the *ANSYS Low-Frequency Electromagnetic Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>Planewav0>Define Wave
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>Planewav0>Wave Status
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>Planewav0>Define Wave
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>Planewav0>Wave Status

---

PMAP, *FORM*, *DISCON*

Creates mapping of the path geometry by defining path interpolation division points.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

FORM

Defines the mapping method:

UNIFORM

Maps uniform divisions (specified on the *nDiv* argument of the **PATH** command) between specified points. This is the default.

ACCURATE

Map geometry using a small division at the beginning and end of each segment. This gives you accurate derivatives, integrals, tangents, and normals for curves which do not have continuous slopes at the specified points. To create nonuniform divisions, the *nDiv* argument of the **PATH** command must be greater than 2.

DISCON

Sets mapping for discontinuities in the field. The divisions are modified to put a point just before and just after the discontinuity. The valid label is MAT, for a material discontinuity. No discontinuity is the default. Discontinuity mapping involves the NOAV option on the **PDEF** command.

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>Path Options
Main Menu>Preprocessor>Path Operations>Define Path>Path Options

---

PMETH

Specifies "p-Method" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

This command is also valid in SOLUTION.

Menu Paths

Utility Menu>List>Status>p-Method

/PMETH, *Key*, *OPTION*

Activates the p-method solution options in the Graphical User Interface (GUI).

SOLUTION: p-Method

MP ME ST <> <> PR <> <> <> PP ED

Key

ON
: Activates the p-method solution options.

OFF
: Deactivates the p-method solution options (default).

STAT
: Lists the current setting of *Key*.

OPTION
: Activates discipline option if *Key* = ON.

0
: Activates the structural p-method solution options (default).

1
: Activates the electrostatic p-method solution options.

Command Default

The p-method solution options are hidden (not shown) in the GUI.

Notes

When this option is activated, only the options applicable to a p-method solution will be displayed in the GUI. This command is not required outside of the GUI.

Menu Paths

Main Menu>Preferences

PMGTRAN, *Fname*, *FREQ*, *Fcnam1*, *Fcnam2*, *Pcnam1*, *Pcnam2*, *Ecnam1*, *Ccnam1*

Summarizes electromagnetic results from a transient analysis.

POST26: Special Purpose

MP ME ST <> <> <> EM <> <> PP ED

Fname
: File name (8 characters maximum) to which tabular data and plot files will be written. Must be enclosed in single quotes when the command is manually typed in. Defaults to MG_TRNS. The data file extension is **.OUT** and the plot file extension is .**PLT**.

FREQ
: Frequency of solution output. Defaults to 1. Every *FREQ*th solution on the results file is output.

Fcnam1, *Fcnam2*
Names of element components for force calculation. Must be enclosed in single quotes when the command is manually typed in.

Pcnam1, *Pcnam2*
Names of element components for power loss calculation. Must be enclosed in single quotes when the command is manually typed in.

Ecnam1, *Ccnam1*
Names of element components for energy and total current calculations, respectively. Must be enclosed in single quotes when the command is manually typed in.

Notes

PMGTRAN invokes an ANSYS macro which calculates and summarizes electromagnetic results from a transient analysis. The results are summarized by element components and listed on the screen as well as written to a file (***Fname*.OUT**). Also, graph plots of results as a function of time are created and written to a file (***Fname*.PLT**) for use in the DISPLAY program.

Two components may be selected for the summary of electromagnetic forces (see **FMAGSUM**), two for power loss, and one each for stored energy (see **SENERGY**) and total current (see **CURR2D**). See the referenced commands for other restrictions.

PMGTRAN is restricted to MKSA units.

Menu Paths

Main Menu>TimeHist Postpro>Elec&Mag>Magnetics

PMLOPT, *ESYS*, *Lab*, *Xminus*, *Xplus*, *Yminus*, *Yplus*, *Zminus*, *Zplus*

Defines perfectly matched layers (PMLs) for a high-frequency analysis.

PREP7: Perfectly Matched Layers

MP <> <> <> <> <> <> EH <> PP ED

ESYS
Element coordinate system number. *ESYS* may be 0 (global Cartesian) or any previously defined local Cartesian coordinate system number (>10). Defaults to 0.

Lab
Label defining the number of dimensions:

ONE
A one-dimensional PML region.

THREE
A 3-D PML region (default).

Xminus
For harmonic analysis, normal reflection coefficient in negative X direction of ESYS. Defaults to 1.E 2.5 (equivalent to -50 dB). For modal analysis, the attenuation factor in negative X direction of ESYS. Defaults to 4.0.

Xplus
For harmonic analysis, normal reflection coefficient in positive X direction of ESYS. Defaults to 1.E 2.5 (equivalent to -50 dB). For modal analysis, the attenuation factor in positive X direction of ESYS. Defaults to 4.0.

Yminus
For harmonic analysis, normal reflection coefficient in negative Y direction of ESYS. Defaults to 1.E 2.5 (equivalent to -50 dB). For modal analysis, the attenuation factor in negative Y direction of ESYS. Defaults to 4.0.

Yplus
For harmonic analysis, normal reflection coefficient in positive Y direction of ESYS. Defaults to 1.E 2.5 (equivalent to -50 dB). For modal analysis, the attenuation factor in positive Y direction of ESYS. Defaults to 4.0.

Zminus
For harmonic analysis, normal reflection coefficient in negative Z direction of ESYS. Defaults to 1.E 2.5 (equivalent to -50 dB). For modal analysis, the attenuation factor in negative Z direction of ESYS. Defaults to 4.0.

Zplus
For harmonic analysis, normal reflection coefficient in positive Z direction of ESYS. Defaults to 1.E 2.5 (equivalent to -50 dB). For modal analysis, the attenuation factor in positive Z direction of ESYS. Defaults to 4.0.

Notes

PMLOPT defines perfectly matched layers (PML) for a high-frequency analysis. Each PML region must have a uniquely defined element coordinate system. Normal reflection coefficient values for a harmonic analysis must be less than 1. Attenuation factor values for a modal analysis must be greater than 1.

Issue **PMLOPT**,STAT to list the current normal reflection coefficient or attenuation factor settings for a PML region. Issue **PMLOPT**,CLEAR to clear all normal reflection coefficient or attenuation factor settings and restore them to the defaults. Issue **PMLOPT**,ESYS,CLEAR to clear all normal reflection coefficient or attenuation factor settings for this element coordinate system and restore them to the defaults.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PMLOpt>Clear
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PMLOpt>Define
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PMLOpt>Status
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PMLOpt>All
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PMLOpt>On ESYS
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PMLOpt>Clear
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PMLOpt>Define
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PMLOpt>Status
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PMLOpt>All
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PMLOpt>On ESYS

PMLSIZE, *FREQB*, *FREQE*, *DMIN*, *DMAX*, *THICK*, *ANGLE*

Determines number of PML layers.

PREP7: Perfectly Matched Layers

MP <> <> <> <> <> <> EH <> PP ED

FREQB
Minimum operating frequency

FREQE
Maximum operating frequency

DMIN
Minimum distance from radiation source to PML interface.

DMAX
Maximum distance from radiation source to PML interface.

THICK
Thickness of PML region. Defaults to 0.

ANGLE
Incident angle of wave to the PML interface. Defaults to 0.

Notes

PMLSIZE determines the number of PML layers for acceptable numerical accuracy.

PMLSIZE must be issued before any meshing commands. If the thickness of the PML region is known, it determines an element edge length (h) and issues ESIZE,h. If the thickness of the PML region is unknown, it determines the number of layers (n) and issues ESIZE,,n.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>PML

PMOPTS, *TOLER*

Defines percentage tolerance for a p-Method solution.

SOLUTION: p-Method

MP ME ST DY <> PR EM <> FL PP ED

TOLER
Percentage tolerance used to determine which elements to fix at the current p-level. If an element's strain energy error is less than this tolerance its p-level will be fixed at the current level. Defaults to 5.

Command Default

Tolerance of 5% will be used.

Notes

Defines the p-level tolerance for p-method solutions (*TOLER*). The default value is satisfactory for most solutions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>p-Method>Solution Options
Main Menu>Solution>Load Step Opts>p-Method>Solution Options

/PMORE, --, *X5*, *Y5*, *X6*, *Y6*, *X7*, *Y7*, *X8*, *Y8*

Creates an annotation polygon (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

--
Unused field.

X5
X location for vertex 5 of polygon (-1.0 < X < 2.0).

Y5
Y location for vertex 5 of polygon (-1.0 < Y < 1.0).

X6
X location for vertex 6 of polygon (-1.0 < X < 2.0).

Y6
Y location for vertex 6 of polygon (-1.0 < Y < 1.0).

X7
X location for vertex 7 of polygon (-1.0 < X < 2.0).

Y7
Y location for vertex 7 of polygon (-1.0 < Y < 1.0).

X8
X location for vertex 8 of polygon (-1.0 < X < 2.0).

Y8
Y location for vertex 8 of polygon (-1.0 < Y < 1.0).

Notes

Defines the 5th through 8th vertices of an annotation polygon [**/POLYGON**]. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

PNGR, *Kywrd*, *OPT*, *VAL*

Provides PNG file export for ANSYS displays.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Kywrd

Specifies various PNG file export options.

COMP

If *Kywrd* = COMP, then *OPT* is either ON or OFF (blank is interpreted as OFF). This option allows you to turn PNG file compression ON or OFF. If OPT = ON, then The VAL field is read to determine the degree of compression. See the VALUE argument for acceptable compression values.

ORIENT

If *Kywrd* = ORIENT, then *OPT* will determine the orientation of the entire plot. *OPT* can be either Horizontal (default) or Vertical.

COLOR

If *Kywrd* = COLOR, then *OPT* will determine the color depth of the saved file. *OPT* can be 0, 1, or 2, corresponding to Black and White, Grayscale, and Color (default), respectively.

TMOD

If *Kywrd* = TMOD, then *OPT* will determine the text method. *OPT* can be either 1 or 0, corresponding to bitmap text (default) or line stroke text, respectively.

DEFAULT

If *Kywrd* = DEFAULT, then all of the default values, for all of the Kywrd parameters listed above, are active.

STAT

Shows the current status of PNG file export.

OPT

OPT can have the following names or values, depending on the value for *Kywrd* (see above).

ON, OFF

If *Kywrd* = COMP, the values On and Off control the use of compression. The degree of compression is determined by VAL

Horizontal, Vertical

If *Kywrd* = ORIENT, the terms Horizontal or Vertical determine the orientation of the plot.

0,1,2

If *Kywrd* = COLOR, the numbers 0, 1, and 2 correspond to Black and White, Grayscale and Color, respectively.

1,0

If *Kywrd* = TMOD, the values 1 and 0 determine whether bitmap (1) or stroke text (0) fonts will be used

VAL

VAL is active only when *Kywrd* = COMP, and determines the degree of compression applied to the exported file (see above).

1

Apply the default, optimum value for compression. This value represents the best combination of speed and compression. It varies according to the release level of the ZLIB compression package.

19
Use this value to specify a specific compression level. 1 is the lowest compression level (fastest) and 9 is the highest compression level (slowest).

Menu Paths

Utility Menu>PlotCtrls>Redirect Plots
Utility Menu>PlotCtrls>Hard Copy

/PNUM, *Label*, *KEY*

Controls entity numbering/coloring on plots.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

Label
Type of numbering/coloring:

NODE
Node numbers on node and element plots.

ELEM
Element numbers and colors on element plots.

DOMAIN
Domains set via **DECOMP**.

SEC
Section numbers and colors on element plots.

MAT
Material set numbers and colors on element and solid model plots (see Notes).

TYPE
Element type reference numbers and colors on element and solid model plots (see Notes).

REAL
Real constant set numbers and colors on element and solid model plots (see Notes).

ESYS
Element coordinate system numbers on element and solid model plots (see Notes).

PART
Element part numbers and colors on element plots (applicable to ANSYS LS-DYNA only).

LOC
Location numbers/colors of the element in the solution sequence (the "wavefront order," see **WAVES** command) on element plots.

Note — LOC and ELEM numbers will be the same unless the model has been reordered.

KP
Keypoint numbers on solid model plots.

LINE
Line numbers on solid model plots (both numbers and colors on line plots).

AREA
Area numbers on solid model plots (both numbers and colors on area plots).

VOLU
Volume numbers on solid model plots (both numbers and colors on volume plots).

SVAL
Stress (or any contour) values on postprocessing plots, and surface load values and colors on model plots (when surface load symbols are on [**/PSF**]). For tabular boundary conditions, the table-evaluated values will be displayed on node, element, or contour displays in POST1 when load symbols (**/PBF**, **/PSF**, **/PBC**) are on and TABNAM is OFF.

TABNAM
Table names for tabular boundary conditions. If this label is turned on, the table name appears next to the appropriate symbol, arrow, face outline, or contour as dictated by the **/PSF**, **/PBC**, and **/PBF** commands.

STAT
Shows current settings for **/PNUM**.

DEFA
Resets all **/PNUM** specifications back to default.

KEY
Switch:

0
Turns OFF numbers/colors for specified label.

1
Turns ON numbers/colors for specified label.

Notes

This command specifies the entity numbering and coloring to be applied to subsequent plots.

The MAT, TYPE, REAL, and ESYS labels activate both the numbering and coloring of the corresponding attributes for elements on **EPLOT**s, keypoints on **KPLOT**s, lines on **LPLOT**s, areas on **APLOT**s, and volumes on ***VPLOT**s.

The ELEM, MAT, TYPE, REAL, ESYS, PART (ANSYS LS-DYNA only), and LOC labels are mutually exclusive, i.e., only one can be specified at a time. Also, turning on a LINE, AREA, or VOLU label will turn off the MAT, TYPE, REAL, and PART labels.

Element and volume numbers are not visible for 3-D elements and volumes when Z-buffering is turned on (**/TYPE**,,[6,7, or 8]).

Use **/PSTATUS** or **/PNUM**,STAT to show settings. Use **/PNUM**,DEFA to reset all specifications back to default. Use the **/NUMBER** command to control whether numbers and colors are displayed together.

/PNUM,LOC and **/PNUM**,ESYS are not supported by PowerGraphics [**/GRAPHICS**,POWER].

/PNUM,DOMAIN,ON or OFF plots preconditioner domains or substructures following auto substructure generation. These domains are used by the DDS, DPCG, or DJCG solvers for preconditioning the solution.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Numbering

POINT
Specifies "Point flow tracing settings" as the subsequent status topic.

POST1: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Trace Points

POLY
Creates a polygonal area based on working plane coordinate pairs.

PREP7: Primitives

MP ME ST DY <> PR EM <> FL PP ED

Notes

Defines a polygonal area on the working plane. The area will be defined with NPT keypoints and NPT lines, where NPT (must be at least 3) is the number of coordinate pairs defined with the **PTXY** command. See the **RPOLY** and **RPR4** commands for other ways to create polygons.

Menu Paths

This command cannot be accessed from a menu.

/POLYGON, *NVERT*, *X1*, *Y1*, *X2*, *Y2*, *X3*, *Y3*, *X4*, *Y4*
Creates annotation polygons (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

NVERT

Number of vertices of polygon ($3 \leq$ *NVERT* ≤ 8). Use **/PMORE** for polygons with more than 4 vertices.

X1
X location for vertex 1 of polygon (-1.0 < X < 2.0).

Y1
Y location for vertex 1 of polygon (-1.0 < Y < 1.0).

X2
X location for vertex 2 of polygon (-1.0 < X < 2.0).

Y2
Y location for vertex 2 of polygon (-1.0 < Y < 1.0).

X3
X location for vertex 3 of polygon (-1.0 < X < 2.0).

Y3
Y location for vertex 3 of polygon (-1.0 < Y < 1.0).

X4
X location for vertex 4 of polygon (-1.0 < X < 2.0).

Y4
Y location for vertex 4 of polygon (-1.0 < Y < 1.0).

Notes

Creates annotation polygons to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All polygons are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** and the **/PSPEC** command to set the attributes of the polygon. Use the **/PMORE** command to define the 5th through 8th vertices of the polygon.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

POPT, *Lop1*

Selects the piping analysis standard for a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

Lop1
Option label:

B31.1
for ANSI B31.1.

NC
for ASME Section III NC, Class 2.

Command Default

ANSI B31.1.

Notes

Selects the piping analysis standard for a piping run [**RUN**]. Affects only the flexibility and stress intensification factors applied to the curved pipe elements.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Specifications

---

/POST1

Enters the database results postprocessor.

SESSION: Processor Entry
POST1: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Notes

Enters the general database results postprocessor (POST1). All load symbols (**/PBC**, **/PSF**, or **/PBF**) are automatically turned off with this command.

This command is valid only at the Begin Level.

Menu Paths

Main Menu>General Postproc

/POST26

Enters the time-history results postprocessor.

SESSION: Processor Entry
POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Notes

Enters the time-history results postprocessor (POST26).

This command is valid only at the Begin Level.

Menu Paths

Main Menu>TimeHist Postpro

POUTRES, *Item1*, *Item2*, *Item3*, -- *Item12*

Controls the nodal DOF and computed element results graphics data that is written to the PGR file.

SOLUTION: Misc Loads

MP ME ST DY <> PR EM <> FL PP ED

Item1, *Item2*, *Item3*,*Item12*

Data written to the PGR file. Acceptable values are as follows:

ALL
: All solution items are written to the PGR file.

NONE
: No solution results are written to the PGR file (default)

NSOL
: Nodal DOF solution (always written)

S
: Stress

NL
: Structural nonlinear data

CONT
: Contact data

EPTO
: Total strain

EPEL
: Elastic Strain

EPTH
: Thermal strain

EPPL
: Plastic Strain

EPCR
: Creep strain

TG
: Thermal gradient

TF
: Thermal flux

EF
: Electric field

D
: Electric flux density

HF
: Magnetic field intensity

B
: Magnetic flux density

FMAG
: Magnetic forces

PG
Pressure gradient

BFE
Body temperatures

TOPO
Densities for topological optimization

Command Default

No solution results are written to the PGR file.

Notes

PowerGraphics data using the CONT option is not available for 2-D models.

This command is also valid in POST1.

Menu Paths

Main Menu>General Postproc>Write PGR File

POWERH

Calculates the rms power loss in a conductor or lossy dielectric.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

Notes

POWERH invokes an ANSYS macro which calculates the time-averaged (rms) power loss in a conductor or lossy dielectric material from a harmonic analysis. The power loss is stored in the parameter PAVG. Conductor losses include solid conductors and surface conductors approximated by impedance or shielding boundary conditions. The power loss density for solid conductors or dielectrics is stored in the element table with the label PLOSSD and may be listed [**PRETAB**] or displayed [**PLETAB**]. PLOSSD does not include surface losses. The elements of the conducting region must be selected before this command is issued. **POWERH** is valid for 2-D and 3-D analyses.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Element Based>Power
Main Menu>General Postproc>Elec&Mag Calc>Element Based>Power Loss

---

PPATH, *POINT*, *NODE*, *X*, *Y*, *Z*, *CS*

Defines a path by picking or defining nodes, or locations on the currently active working plane, or by entering specific coordinate locations.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

POINT
The point number. It must be greater than zero and less than or equal to the *nPts* value specified on the **PATH** command if graphical picking is not being used.

NODE
The node number defining this point. If blank, use the X, Y, Z coordinates to define the point. A valid node number will override *X*, *Y*, *Z* coordinate arguments.

PICK
Pick the node.

(blank)
Use the X, Y, Z coordinates to define the point.

X, *Y*, *Z*
X, *Y*, *Z* arguments specify the location of the point in the global Cartesian coordinate system. Use this argument only if you omit the *NODE* argument.

PICK
Pick the point.

CS
The coordinate system for interpolation of the path between the previous point and this point. Omit this argument if you wish to use the currently active (**CSYS**) coordinate system. If the coordinate system of two adjacent points is different, the *CS* value of the latter point will be used.

Notes

For linearized stress calculations, the path must be defined with nodes.

For information on displaying paths you have defined, see Defining Data to be Retrieved in the *ANSYS Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>By Location
Main Menu>General Postproc>Path Operations>Define Path>By Nodes
Main Menu>General Postproc>Path Operations>Define Path>Modify Path
Main Menu>General Postproc>Path Operations>Define Path>On Working Plane
Main Menu>Preprocessor>Path Operations>Define Path>By Location
Main Menu>Preprocessor>Path Operations>Define Path>By Nodes
Main Menu>Preprocessor>Path Operations>Define Path>Modify Path
Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane

PPLOT

Displays an element plot indicating each element's final p-level.

POST1: Results

MP ME ST DY <> PR <> <> <> PP ED

Notes

The final polynomial level (an integer value ranging from 2 to 8) is displayed at the center of the element's facets [**/EFACET**] in a p-method analysis.

Menu Paths

Main Menu>General Postproc>Plot Results>p-Method>p-Levels

PPRANGE, *START*, *MAX*

Specifies a range of p-level values for use in a p-method solution.

SOLUTION: p-Method

MP ME ST <> <> PR <> <> <> PP ED

START
Starting p-level. Defaults to 2.

MAX
Maximum p-level. Defaults to 8.

Command Default

As described for each argument above.

Notes

The minimum possible p-level is 2, and the maximum possible p-level is 8. This command sets a global p-range for the entire model, where KEYOPT settings [**ET**] control individual element p-levels. Element KEYOPT settings override **PPRANGE** settings.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>p-Method>Set p range
Main Menu>Solution>Load Step Opts>p-Method>Set p range

PPRES, *PRESS*

Defines the internal pressure for a piping run.

PREP7: Piping

MP ME ST DY <> PR <> <> <> PP ED

PRESS

Pipe internal pressure.

Notes

Defines the pipe internal pressure for a piping run [**RUN**]. These pressures are assigned to the elements as they are generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Loads

PRANGE, *LINC*, *VMIN*, *VMAX*, *XVAR*

Determines the path range.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

LINC, *VMIN*, *VMAX*

Set the range for listing or displaying the table locations between a minimum value (*VMIN*) and a maximum value (*VMAX*) of the path distance with a location increment of *LINC* (defaults to 1). The first location begins at *VMIN*.

XVAR

Path variable item to be used as the x-axis plot variable. Any valid path variable may be used (**PDEF** command). Default variable is the path distance, S.

Command Default

Include every interpolation point and entire path distance.

Notes

Determines the path distance range for use with the **PRPATH** and **PLPATH** commands.

Menu Paths

Main Menu>General Postproc>Path Operations>Plot Path Item>Path Range

---

PRCONV

Lists convergence values versus characteristic p-level.

POST1: Results

MP ME ST DY <> PR <> <> <> PP ED

Notes

After a p-method solution, this command lists in tabular form all convergence values versus characteristic p-level.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>General Postproc>List Results>p-Convergence

PRCPLX, *KEY*

Defines the output form for complex variables.

POST26: Listing

MP ME ST DY <> PR EM <> <> PP ED

KEY

Output form key:

0

Real and imaginary parts.

1

Amplitude and phase angle. Stored real and imaginary data are converted to amplitude and phase angle upon output. Data remain stored as real and imaginary parts.

Notes

Defines the output form for complex variables. Used only with harmonic analyses (**ANTYPE**,HARMIC).

All results data are stored in the form of real and imaginary components and converted to amplitude and/or phase angle as specified via the **PRCPLX** command. The conversion is not valid for derived results (such as principal stress/strain, equivalent stress/strain and USUM).

Menu Paths

Main Menu>TimeHist Postpro>Settings>List

PRECISION, *LABEL*

Specifies machine precision for solvers (currently valid only for PCG solvers).

SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> <> PP ED

LABEL
Machine precision to use for solvers:

0 or double
Specifies double precision (default).

1 or single
Specifies single precision.

Notes

When the PCG or PCGOUT solvers are specified by the **EQSLV** command, this command can be used to specify single precision for the solver. By using single precision, about 30% less in-core memory is used by the PCG solver. However, using single precision can lead to unconvergence for the PCG solver when the problem is ill-conditioned. Double precision is the default and the recommended option for the PCG solvers

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

PRED, *Sskey*, *--*, *Lskey*

Activates a predictor in a nonlinear analysis.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR <> <> <> PP ED

Sskey
Substep predictor key:

OFF
No prediction is done (default if rotation DOF are present or element 65 is present).

ON
Use a predictor on all substeps after the first (default, unless rotation DOF are present, or SOLID65 is present).

--
Unused field.

Lskey
Load step predictor:

OFF
No prediction across load steps is done (default).

ON
Use a predictor also on the first substep of the load step (*Sskey* must also be ON).

Command Default

Sskey is ON if there is no rotation DOF present in the model. If rotation DOF are present, or if SOLID65 is present, *Sskey* is OFF.

Notes

Activates a predictor in a nonlinear analysis on the degree of freedom solution for the first equilibrium iteration of each substep.

The default values given for this command assume **SOLCONTROL**,ON (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL**,ON and **SOLCONTROL**,OFF.

You cannot use the DOF solution predictor [**PRED**], automatic time stepping [**AUTOTS**], or line search [**LNSRCH**] with the arc-length method [**ARCLEN**, **ARCTRM**]. If you activate the arc-length method after you set **PRED**, **AUTOTS**, or **LNSRCH**, a warning message appears. If you choose to proceed with the arc-length method activation, ANSYS disables your DOF predictor, automatic time stepping, and line search settings.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Predictor
Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Solution>Load Step Opts>Nonlinear>Predictor

/PREP7

Enters the model creation preprocessor.

SESSION: Processor Entry
PREP7: Database

MP ME ST DY <> PR EM <> FL PP ED

Notes

Enters the general input data preprocessor (PREP7).

This command is valid only at the Begin Level.

Menu Paths

Main Menu>Preprocessor

PRERR

Prints SEPC and TEPC.

POST1: Listing

MP ME ST DY <> PR <> <> <> PP ED

Notes

Prints the percent error in structural energy norm (SEPC) and the thermal energy norm percent error (TEPC). Approximations of mesh discretization error associated with a solution are calculated for analyses having structural or thermal degrees of freedom.

The structural approximation is based on the energy error (which is similar in concept to the strain energy) and represents the error associated with the discrepancy between the calculated stress field and the globally continuous stress field (see POST1 - Error Approximation Technique in the *ANSYS, Inc. Theory Reference*). This discrepancy is due to the assumption in the elements that only the displacements are continuous at the nodes. The stress field is calculated from the displacements and should also be continuous, but generally is not. Structural analyses should be linear elastic and may use solid elements having only structural degrees of freedom (excluding SOLID46, SOLID65, HYPER56, HYPER58, HYPER74, HYPER84, HYPER86, HYPER158 VISCO88, VISCO106, VISCO107, and VISCO108) and 3-D shell elements (excluding SHELL28, SHELL41, SHELL91, and SHELL99).

Thermal analyses may use any solid and shell thermal element having only temperature degrees of freedom. The thermal approximation is based on the total heat flow dissipation and represents the error associated with the discrepancy between the calculated nodal thermal flux within an element and a continuous global thermal flux. This continuous thermal flux is calculated with the normal nodal averaging procedure.

The volume (result label VOLU) is used to calculate the energy error per element (result label SERR for the structural energy error and TERR for the thermal energy error). These energy errors, along with the appropriate energy, are then used to calculate the percent error in energy norm (SEPC for structural and TEPC for thermal). These percentages can be listed by the **PRERR** command, retrieved by the ***GET** command (with labels SEPC and TEPC) for further calculations, and shown on the displacement display [**PLDISP**], as applicable.

For structural analyses, the maximum absolute value of nodal stress variation of any stress component for any node of an element (result item SDSG) is also calculated. Similarly, for thermal gradient components, TDSG is calculated. Minimum and maximum result bounds considering the possible effect of discretization error will be shown on contour displays [**PLNSOL**]. For shell elements, the top surface location is used to produce a meaningful percentage value. SERR, TERR, SEPC, TEPC, SDSG, and TDSG will be updated whenever the nodal stresses or fluxes are recalculated.

If the energy error is a significant portion of the total energy, then the analysis should be repeated using a finer mesh to obtain a more accurate solution. The energy error is relative from problem to problem but will converge to a zero energy error as the mesh is refined. An automated adaptive meshing procedure using this energy error is described with the **ADAPT** macro.

Menu Paths

Main Menu>General Postproc>List Results>Percent Error
Utility Menu>List>Results>Percent Error

PRESOL, *Item*, *Comp*

Prints the solution results for elements.

POST1: Results

MP ME ST DY <> PR EM <> <> PP ED

Item

Label identifying the item. Valid item labels are shown in PRESOL - Valid Item and Component Labels below. Some items also require a component label.

Comp

Component of the item (if required). Valid component labels are shown in PRESOL - Valid Item and Component Labels below.

Notes

Prints the solution results for the selected elements in the sorted sequence. For example, **PRESOL**,S prints the stress items SX, SY, SZ, SXY, SYZ, and SXZ for the node locations of the element. Component results are in the global Cartesian coordinate directions unless transformed [**RSYS**]. Shell elements print values at the top, then bottom of the element (or layer). If KEYOPT(8) = 2 (for SHELL93, SHELL181, SHELL208, or SHELL209) or KEYOPT(11) = 2 (SHELL63), then the results are printed in the order TOP, BOT and then MID of each element, (or layer). The MID value will be the actual value as written to the results file. Items are listed as columns of a table versus element number. An exception occurs for item ELEM which uses an element format (all applicable line element results are listed per element) instead of a tabular format. The **FORCE** command can be used to define which component of the nodal load is to be used (static, damping, inertia, or total). See the **ETABLE** and **PRETAB** commands for printing items not available through this command (such as line element results).

For PowerGraphics [**/GRAPHICS**,POWER], results are listed only for the element surface. The items marked with [1] are not supported by PowerGraphics.

PRESOL - Valid Item and Component Labels

Item	Comp	Description
	Valid item and component labels for element results are:	
S		Component (X, Y, Z, XY, YZ, XZ) stresses.
EPEL		Component (X, Y, Z, XY, YZ, XZ) elastic strains.
EPTH		Component (X, Y, Z, XY, YZ, XZ) thermal strains.
EPPL		Component (X, Y, Z, XY, YZ, XZ) plastic strains.
EPCR		Component (X, Y, Z, XY, YZ, XZ) creep strains.
EPSW		Swelling strain.
EPTO		Component (X, Y, Z, XY, YZ, XZ) total mechanical strains (EPEL + EPPL + EPCR).
EPTT		Component (X, Y, Z, XY, YZ, XZ) total mechanical *and* thermal strains (EPEL + EPPL + EPCR + EPTH).
NL		Nonlinear items (SEPL, SRAT, HPRES, EPEQ, CREQ, PSV, PLWK).
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
SVAR	1,2,3, ... N	State variable.

Item	Comp	Description
GKS		Gasket component (X, XY, XZ) stress.
GKD		Gasket component (X, XY, XZ) total closure.
GKDI		Gasket component (X, XY, XZ) total inelastic closure.
GKTH		Gasket component (X, XY, XZ) thermal closure.
CONT		Contact items (STAT, PENE, PRES, SFRIC, STOT, SLIDE, GAP, FLUX). See component descriptions in **PLNSOL**.
TG		Component (X, Y, Z) thermal gradients and vector sum (SUM).
TF		Component (X, Y, Z) thermal fluxes and vector sum (SUM).
PG		Component (X, Y, Z) pressure gradients and vector sum (SUM).
EF		Component (X, Y, Z) electric fields and vector sum (SUM).
D		Component (X, Y, Z) electric flux densities and vector sum (SUM).
H		Component (X, Y, Z) magnetic field intensities and vector sum (SUM).
B		Component (X, Y, Z) magnetic flux densities and vector sum (SUM).
FMAG		Component (X, Y, Z) magnetic forces and vector sum (SUM) [1].
P		Poynting vector components (X, Y, Z) and sum (SUM) [1].
F		Component (X, Y, Z) structural forces. Use **FORCE** for type [1].
M		Component (X, Y, Z) structural moments. Use **FORCE** for type [1].
HEAT		Heat flow. Use **FORCE** for type [1].
FLOW		Fluid flow. Use **FORCE** for type.
AMPS		Current flow [1]. Use **FORCE** for type.
CHRG		Charge [1]. Use **FORCE** for type.
FLUX		Magnetic flux [1].
VF		Component (X, Y, Z) fluid "forces". Use **FORCE** for type [1].
CSG		Component (X, Y, Z) magnetic current segments. Use **FORCE** for type [1].
FORC		All available force items (F to CSG above). (10 maximum) [1].
BFE		Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
ELEM		All applicable element results (available only for structural line elements) [1].
SERR		Structural error energy [1].
SDSG		Absolute value of the maximum variation of any nodal stress component [1].
TERR		Thermal error energy [1].
TDSG		Absolute value of the maximum variation of any nodal thermal gradient component [1].
SENE		"Stiffness" energy or thermal heat dissipation. Same as TENE [1].
TENE		Thermal heat dissipation or "stiffness" energy. Same as SENE [1].
KENE		Kinetic energy [1].
JHEAT		Element Joule heat generation (coupled-field calculation) [1].

Item	Comp	Description
JS		Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low frequency electric analyses. Components (X, Y, Z) and vector sum (SUM). [1].
JT		Total measureable current density in low-frequency electromagnetic analyses. (Conduction current density in a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM). [1].
JC		Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM). [1].
MRE		Magnetic Reynolds number [1].
VOLU		Volume of volume element [1].
CENT		Centroid X, Y, or Z location (based on shape function) in the active coordinate system [1].
LOCI		Integration point location
SMISC	*snum*	Element summable miscellaneous data value at sequence number *snum* (Shown in the Output Data section of each element description. See Chapter 4 of the *ANSYS Elements Reference*) [1].
NMISC	*snum*	Element nonsummable miscellaneous data value at sequence number *snum* (Shown in the Output Data section of each element description. See Chapter 4 of the *ANSYS Elements Reference*) [1].
TOPO		Densities used for topological optimization. This applies to the following types of elements: PLANE2, PLANE82, SOLID92, SHELL93, SOLID95.

1. Not supported by PowerGraphics

Menu Paths

Main Menu>General Postproc>List Results>Element Solution
Utility Menu>List>Results>Element Solution

PRETAB, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*

Prints the element table items.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

Lab1, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*

Print selected items. Valid labels are (blank) or any label as specified with the **ETABLE** command. Convenience labels may be used for *Lab1* to select groups of labels (10 labels maximum): GRP1 for first 10 stored items; GRP2 for items 11 to 20; GRP3 for items 21 to 30; GRP4 for items 31 to 40; GRP5 for items 41 to 50. Enter **ETABLE**,STAT command to list stored item order. If all labels are blank, print first 10 stored items (GRP1).

Notes

Prints the items stored in the table defined with the **ETABLE** command. Item values will be listed for the selected elements in the sorted sequence [**ESORT**]. The **FORCE** command can be used to define which component of the nodal load is to be used (static, damping, inertia, or total).

Portions of this command are not supported by PowerGraphics [**/GRAPHICS**,POWER].

Menu Paths

Main Menu>General Postproc>Element Table>List Elem Table
Main Menu>General Postproc>List Results>Elem Table Data
Utility Menu>List>Results>Element Table Data

PRHFFAR, *Opt*, *Lab*, *PHI1*, *PHI2*, *NPHI*, *THETA1*, *THETA2*, *NTHETA*, *RADIUS*, *CS*, *Comp*
Prints electromagnetic far fields and far field parameters.

POST1: Special Purpose

MP <> <> <> <> <> <> EH <> PP ED

Opt
Print option:

FIELD
Electromagnetic far field.

RCS
Radar cross section.

RCSN
Normalized radar cross section.

PATT
Antenna pattern.

DGAIN
Antenna directive gain.

PRAD
Antenna radiation power.

PGAIN
Antenna power gain.

EFF
Antenna radiation efficiency.

Lab
As shown below, data entered in the Lab field will vary, depending on the print option (Opt).

Valid Lab Data Labels

Opt	*Lab*
FIELD	EF -- Electric field (default) H -- Magnetic field

Opt	*Lab*
RCS and RCSN	NONE -- Radar echo area (default) PP -- Phi-Phi polarization PT -- Phi-Theta polarization TP -- Theta-Phi polarization TT -- Theta-Theta polarization
PATT	RECT -- Radiation pattern (default)
DGAIN	MAX -- Directivity (default) RECT -- Directive gain
PRAD	Ignore Lab.
PGAIN	VALUE -- Input power (watts)
EFF	VALUE -- Input power (watts)

The following *PHI1*, *PHI2*, *NPHI*, *THETA1*, *THETA2*, and *NTHETA* arguments are used only with *Opt* = FIELD, RCS, RCSN, PATT, or *Opt* = FIELD, *Lab* = RECT.

PHI1, *PHI2*
Starting and ending ϕ angles (degrees) in the spherical coordinate system. Defaults to 0.

NPHI
Number of divisions between the starting and ending ϕ angles for data computations. Defaults to 0.

THETA1, *THETA2*
Starting and ending θ angles (degrees) in the spherical coordinate system. Defaults to 0.

NTHETA
Number of divisions between the starting and ending θ angles for data computations. Defaults to 0.

The following arguments are used only with *Opt* = FIELD.

RADIUS
Radius for far field. Defaults to 1.

CS
Coordinate system type:

0
Cartesian (default).

2
Spherical.

Comp
Coordinate system component:

SUM
Magnitude of electromagnetic field (default).

X
X-component (CS = 0) or radial component (CS = 2).

Y
Y-component (CS = 0) or ϕ component (CS = 2).

Z
Z-component (CS = 0) or θ component (CS = 2).

See Spherical Coordinates in the *ANSYS Low-Frequency Electromagnetic Analysis Guide*.

Notes

PLHFFAR prints electromagnetic far fields and far field parameters as determined by the equivalent source principle. Use this command to print far electromagnetic field, radar cross section, antenna radiation pattern, antenna directive gain, antenna directivity, antenna radiation power, antenna power gain, or antenna radiation efficiency.

Menu Paths

Main Menu>General Postproc>List Results>Field Extension>Direct Gain
Main Menu>General Postproc>List Results>Field Extension>Efficiency
Main Menu>General Postproc>List Results>Field Extension>Far Field
Main Menu>General Postproc>List Results>Field Extension>Pattern
Main Menu>General Postproc>List Results>Field Extension>Power Gain
Main Menu>General Postproc>List Results>Field Extension>Rad Power
Main Menu>General Postproc>List Results>Field Extension>RCS
Main Menu>General Postproc>List Results>Field Extension>RCS Normalized

---

PRI2, *P51X*, *Z1*, *Z2*

Creates a polygonal area or a prism volume by vertices (GUI).

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

Notes

Creates a polygonal area or a prism volume using the vertices as input. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if graphical picking is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

For polygons, the **PRI2** command will appear in the log file as **PRI2**,*P51X*,0.0,0.0 preceded by **FITEM** commands that define the vertices (in global Cartesian coordinates). For prisms, the **PRI2** command will appear in the log file as **PRI2**,*P51X* preceded by **FITEM** commands that define the vertices and the Z-end of the prism.

See the **RPOLY**, **POLY**, **RPRISM**, **PRISM**, and **RPR4** commands for other ways to create polygons and prisms.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Vertices
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Vertices

---

PRIM

Specifies "Solid model primitives" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Solid Model

PRINT

Specifies "Print settings" as the subsequent status topic.

POST1: Status
POST26: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>List Results
Utility Menu>List>Status>TimeHist Postproc>List

PRISM, *Z1*, *Z2*

Creates a prism volume based on working plane coordinate pairs.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

Z1, *Z2*
Working plane Z coordinates of the top and bottom of the prism.

Notes

Defines a prism volume based on the working plane. The top and bottom areas will each be defined with NPT keypoints and NPT lines, where NPT (must be at least 3) is the number of coordinate pairs defined with **PTXY** command. Also, a line will be defined between each point pair on the top and bottom face. See the **RPRISM** and **RPR4** commands for other ways to create prisms.

Menu Paths

This command cannot be accessed from a menu.

PRITER
Prints solution summary data.

POST1: Listing

MP ME ST DY <> PR EM <> FL PP ED

Notes

Prints solution summary data (such as time step size, number of equilibrium iterations, convergence values, etc.) from a static or full transient analysis. All other analyses print zeros for the data.

Menu Paths

Main Menu>General Postproc>List Results>Iteration Summry
Utility Menu>List>Results>Iteration Summry

PRJSOL, *Label*, *Comp*
Print joint element output.

POST1: Results

MP ME ST <> <> PR <> <> <> PP ED

Label
Label identifying the item. Valid in POST1 only. Some items also require a component label.

REAC -- reaction forces and moments.
DISP - displacements.
ROT - rotations.
SMISC -- summable miscellaneous quantities.

Comp
Component of the item (if required). For DISP and ROT, enter the direction, x, y, or z. For SMISC enter a valid number.

Notes

Valid for MPC184.

Menu Paths

This command cannot be accessed from a menu.

PRNLD, *Lab*, *TOL*, *Item*

Prints the summed element nodal loads.

POST1: Results

MP ME ST DY <> PR EM <> <> PP ED

Lab

Nodal reaction load type. If blank, use the first ten of all available labels. Valid labels are: Structural force labels: FX, FY or FZ (forces); F (includes FX, FY and FZ); MX, MY or MZ (moments); M (includes MX, MY and MZ). Thermal force labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow). Fluid force labels: FLOW (fluid flow); VFX, VFY and VFZ (fluid "forces"); VF (includes VFX, VFY and VFZ). Electric force labels: AMPS (current flow); CHRG (charge); CURT (current); VLTG (voltage drop). Magnetic force labels: FLUX (magnetic flux); CSGX, CSGY, CSGZ (magnetic current segments); CSG (includes CSGX, CSGY and CSGZ).

TOL

Tolerance value about zero within which loads are not printed. Defaults to 1.0E-9 times the absolute value of the maximum load on the selected nodes. If zero, print all nodal loads.

ITEM

Selected set of nodes.

(blank)
: Prints the summed element nodal loads for all selected nodes (default), excluding contact elements.

CONT
: Prints the summed element nodal loads for contact nodes only.

BOTH
: Prints the summed element nodal loads for all selected nodes, including contact nodes.

Notes

Prints the summed element nodal loads (forces, moments, heat flows, flux, etc.) for the selected nodes in the sorted sequence. Results are in the global Cartesian coordinate directions unless transformed [**RSYS**]. Zero values (within a tolerance range) are not printed. Loads applied to a constrained degree of freedom are not included. The **FORCE** command can be used to define which component of the nodal load is to be used (static, damping, inertia, or total).

By default, **PRNLD** excludes elements TARGE169 - CONTA175. **PRNLD**,,,CONT will only account for nodal forces on selected contact elements (CONTA171 - CONTA175). **PRNLD**,,,BOTH will account for nodal forces on all selected nodes, including contact nodes.

Menu Paths

Main Menu>General Postproc>List Results>Nodal Loads
Utility Menu>List>Results>Nodal Loads

PRNSOL, *Item*, *Comp*

Prints the nodal solution results.

POST1: Results

MP ME ST DY <> PR EM <> FL PP ED

Item

Label identifying the item. Valid item labels are shown in PRNSOL - Valid Item and Component Labels below. Some items also require a component label.

Comp

Component of the item (if required). Valid component labels are shown in PRNSOL - Valid Item and Component Labels below. Defaults to COMP.

Notes

Prints the nodal solution results for the selected nodes in the sorted sequence. For example, **PRNSOL**,U,X prints the X component of displacement vector U (i.e., the UX degree of freedom). Component results are in the global Cartesian coordinate directions unless transformed [**RSYS**]. Various element results also depend upon the recalculation method and the selected results location [**AVPRIN**, **RSYS**, **LAYER**, **SHELL**, and **NSEL**]. If **LAYER** is specified, then the resulting output will be listed in Full graphics mode. The **FORCE** command can be used to define which component of the nodal load is to be used (static, damping, inertia, or total).

PowerGraphics can affect your nodal solution listings. If a node is common to more than one element, or if a geometric discontinuity exists, several conflicting listings may result. For instance, a corner node incorporating results from a brick element and three shell elements could yield as many as nine different results. ANSYS does not average result listings when shell elements are present. It is important to analyze the listings at discontinuities to ascertain the significance of each set of data.

For PowerGraphics [**/GRAPHICS**,POWER], results are listed only for the model exterior surface. If **NSORT**, **ESORT** or **/ESHAPE** are specified with PowerGraphics activated, then the **PRNSOL** listings will be the same as in Full graphics mode. The items marked with [2] are not supported by PowerGraphics. To print midside nodes, you must first issue **/EFACET**,2.

PRNSOL - Valid Item and Component Labels

Item	Comp	Description
	Valid item and component labels for nodal degree of freedom results are:	
U	X, Y, Z	X, Y, or Z structural displacement.
"	COMP	X, Y, and Z structural displacements and vector sum.
ROT	X, Y, Z	X, Y, or Z structural rotation.
"	COMP	X, Y, and Z structural rotations and vector sum.
TEMP [1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity in a fluid analysis, or X, Y, or Z nodal velocity in a structural transient dynamic analysis[3] .

Item	Comp	Description
"	COMP	X, Y, and Z fluid velocity and vector sum in a fluid analysis, or X, Y, and Z nodal velocity and vector sum in a structural transient dynamic analysis[3].
A	X, Y, Z	X, Y, or Z magnetic vector potential in an electromagnetic analysis, or X, Y, or Z acceleration in a structural transient dynamic analysis[3].
"	COMP	X, Y, and Z magnetic vector potential and vector sum in an electromagnetic analysis, or X, Y, and Z acceleration and vector sum in a structural transient dynamic analysis[3].
CURR		Current.
EMF		Electromotive force drop.
ENKE		Turbulent kinetic energy.
ENDS		Turbulent energy dissipation.
SP0*n*		Mass fraction of species *n*, where *n* = 1 to 6 (FLOTRAN). If a species is given a user-defined name [**MSSPEC**], use that name instead of SP0*n* [2].
DOF		All available degree of freedom labels (10 maximum).
	Valid item and component labels for element results are:	
S	COMP	X, Y, Z, XY, YZ, and XZ component stresses.
"	PRIN	S1, S2, S3 principal stresses, SINT stress intensity, and SEQV equivalent stress.
"	FAIL	Maximum Stress, Tsai-Wu Strength Index, and inverse of Tsai-Wu Strength Ratio Index Failure Criteria. (Only works if FC command information is provided.) [2]
EPEL	COMP	Component elastic strains.
"	PRIN	Principal elastic strains, strain intensity, and equivalent strain.
"	FAIL	Maximum Strain Failure Criteria. (Only works if FC command information is provided.)
EPTH	COMP	Component thermal strains.
"	PRIN	Principal thermal strains, strain intensity, and equivalent strain.
EPPL	COMP	Component plastic strains.
"	PRIN	Principal plastic strains, strain intensity, and equivalent strain.
EPCR	COMP	Component creep strains.
"	PRIN	Principal creep strains, strain intensity, and equivalent strain.
EPSW		Swelling strain.
EPTO	COMP	Component total mechanical strains (EPEL + EPPL + EPCR).
"	PRIN	Principal total mechanical strains, strain intensity, and equivalent strain.
EPTT	COMP	Component total mechanical *and* thermal strains (EPEL + EPPL + EPCR + EPTH).
"	PRIN	Principal total mechanical *and* thermal strains, strain intensity, and equivalent strain.
NL		Nonlinear items (SEPL, SRAT, HPRES, EPEQ, CREQ, PSV, PLWK).
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.

Item	Comp	Description
SVAR	1, 2, 3, ... N	State variable.
GKS	COMP	X, XY, XZ component gasket stress.
GKD	COMP	X, XY, XZ component gasket total closure.
GKDI	COMP	X, XY, XZ component gasket total inelastic closure.
GKTH	COMP	X, XY, XZ component thermal closure.
CONT		Contact items (STAT[4], PENE, PRES, SFRIC, STOT, SLIDE, GAP, FLUX, CNOS). See component descriptions in **PLNSOL**.
TG	COMP	Component thermal gradients and vector sum.
TF	COMP	Component thermal fluxes and vector sum.
PG	COMP	Component pressure gradients and vector sum.
EF	COMP	Component electric fields and vector sum.
D	COMP	Component electric flux densities and vector sum.
H	COMP	Component magnetic field intensities and vector sum.
B	COMP	Component magnetic flux densities and vector sum.
FMAG	COMP	Component magnetic forces and vector sum [2].
JC	COMP	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM). [2].
BFE		Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
TOPO		Densities used for topological optimization. This applies to nodes attached to the following types of elements: PLANE2, PLANE82, SOLID92, SHELL93, SOLID95.
	Valid item and component labels for FLOTRAN nodal results are:	
TTOT		Total temperature.
HFLU		Heat flux.
HFLM		Heat transfer (film) coefficient.
COND		Fluid laminar conductivity.
PCOE		Pressure coefficient.
PTOT		Total (stagnation) pressure.
MACH		Mach number.
STRM		Stream function. (2-D applications only.)
DENS		Fluid density.
VISC		Fluid laminar viscosity.
SPHT		Specific heat.
EVIS		Fluid effective viscosity.
CMUV		Turbulent viscosity coefficient.
ECON		Fluid effective conductivity.
YPLU		Y+, a turbulent law of the wall parameter.
TAUW		Shear stress at the wall.
SFTS		Surface tension coefficient.
LMD*n*		Laminar mass diffusion coefficient for species *n*, where *n* = 1 to 6.
EMD*n*		Effective mass diffusion coefficient for species *n*, where *n* = 1 to 6.

Item	Comp	Description
RDFL		Radiation heat flux.

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . ., TTOP instead of TEMP.
2. Not supported by PowerGraphics
3. Nodal velocities and nodal accelerations are available in explicit dynamic analyses (LS-DYNA) and in ANSYS structural analyses with **ANTYPE**,TRANS
4. For the CONT items for elements CONTA171 through CONTA175, the reported data is averaged across the element.

Menu Paths

Main Menu>General Postproc>List Results>Nodal Solution
Main Menu>General Postproc>List Results>Sorted Listing>Sort Nodes
Utility Menu>List>Results>Nodal Solution

PROD, *IR*, *IA*, *IB*, *IC*, *Name*, --, --, *FACTA*, *FACTB*, *FACTC*

Multiplies variables.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA, *IB*, *IC*
Reference numbers of the three variables to be operated on. If only two leave *IC* blank. If only one, leave *IB* blank also.

Name
Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--
Unused field

--
Unused field

FACTA, *FACTB*, *FACTC*
Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

Notes

Multiplies variables (up to three at once) according to the operation:

IR = (*FACTA* x *IA*) x (*FACTB* x *IB*) x (*FACTC* x *IC*)

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Multiply

PRPATH, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

Prints path items along a geometry path.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

Lab1, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

Labels identifying the path items to be printed. Up to six items may be printed at a time. Predefined path geometry items XG, YZ, ZG, and S [**PDEF**] may also be printed.

Notes

Prints path items with respect to a geometry path (as defined by the **PATH** and **PPATH** commands). Path items and their labels must have been defined with the **PDEF**, **PVECT**, **PCALC**, **PDOT**, or **PCROSS** commands. Path items may also be displayed with the **PLPATH** and **PLPAGM** commands. See the **PRANGE** command for range control of the path.

Menu Paths

Main Menu>General Postproc>List Results>Path Items
Main Menu>General Postproc>Path Operations>Plot Path Item>List Path Items
Utility Menu>List>Results>Path Data

PRRFOR, *Lab*

Used with the FORCE command. Prints the constrained node reaction solution.

POST1: Results

MP ME ST DY <> PR EM <> <> PP ED

Lab

Nodal reaction load type. If blank, use the first ten of all available labels. Valid labels are: Structural force labels: FX, FY or FZ (forces); F (FX, FY and FZ); MX, MY or MZ (moments); M (MX, MY and MZ). Thermal force labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow). Fluid force labels: FLOW (fluid flow); VFX, VFY and VFZ (fluid "forces"); VF (VFX, VFY and VFZ). Electric force labels: AMPS (current flow); CHRG (charge); CURT (current); VLTG (voltage drop). Magnetic force labels: FLUX (magnetic flux); CSGX, CSGY, CSGZ (magnetic current segments); CSG (CSGX, CSGY and CSGZ); CURT (current), VLTG (voltage drop).

Notes

PRRFOR has the same functionality as the **PRRSOL** command; use **PRRFOR** instead of **PRRSOL** when a **FORCE** command has been issued.

PRRFOR cannot be used when a squaring operation has been carried out on a load case [such as in PSD and spectrum analyses]. **PRRSOL** values are correct in this case, because they are squared after summing the nodal forces.

Menu Paths

This command cannot be accessed from a menu.

PRRSOL, *Lab*

Prints the constrained node reaction solution.

POST1: Results

MP ME ST DY <> PR EM <> <> PP ED

Lab

Nodal reaction load type. If blank, use the first ten of all available labels. Valid labels are: Structural force labels: FX, FY or FZ (forces); F (FX, FY and FZ); MX, MY or MZ (moments); M (MX, MY and MZ); BMOM (bimoments). Thermal force labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow). Fluid force labels: FLOW (fluid flow); VFX, VFY and VFZ (fluid "forces"); VF (VFX, VFY and VFZ). Electric force labels: AMPS (current flow); CHRG (charge); CURT (current); VLTG (voltage drop). Magnetic force labels: FLUX (magnetic flux); CSGX, CSGY, CSGZ (magnetic current segments); CSG (CSGX, CSGY and CSGZ); CURT (current), VLTG (voltage drop).

Notes

Prints the constrained node reaction solution for the selected nodes in the sorted sequence. For coupled nodes and nodes in constraint equations, the sum of all reactions in the coupled or constraint equation set appears at the primary node of the set. Results are in the global Cartesian coordinate directions unless transformed [**RSYS**]. **PRRSOL** is not valid if any load is applied to a constrained node in the direction of the constraint *and* any of the following is true: a) **LCOPER** has been used, or b) **LCASE** has been used to read from a load case file or c) the applied loads and constraints in the database are not the ones used to create the results data being processed.

Use **PRRFOR** instead of **PRRSOL** with the **FORCE** command.

Menu Paths

Main Menu>General Postproc>List Results>Reaction Solu
Utility Menu>List>Results>Reaction Solution

PRSECT, *RHO*, *KBR*

Calculates and prints linearized stresses along a section path.

POST1: Path Operations

MP ME ST DY <> PR <> <> <> PP ED

RHO

In-plane (X-Y) average radius of curvature of the inside and outside surfaces of an axisymmetric section. If zero (or blank), a plane or 3-D structure is assumed. If nonzero, an axisymmetric structure is assumed. Use any large number (or -1) for an axisymmetric straight section.

KBR

Through-thickness bending stresses key for an axisymmetric analysis (*RH0* $\neq$ 0):

0

Include the thickness-direction bending stresses.

1
Ignore the thickness-direction bending stresses.

Notes

Calculates and prints linearized stresses along a section path [**PATH**] as defined by the two points specified with the **PPATH** command. Use only the NODE option on **PPATH** to define the path, which must be two nodes only. The path must be entirely within the selected elements (that is, there must not be any element gaps along the path). No data is retained on the path; the path is used only to retrieve the two path node numbers. Gives component and principal stresses at the beginning, mid-length, and end of the section. Stresses are also separated into membrane, bending, membrane plus bending, peak, and total stress categories. The total stress can also be printed for intermediate points with the **PRPATH** command. Stresses are output in section coordinates (SX along the path, SY normal, and SZ hoop) for axisymmetric sections, and in the active coordinate system [**RSYS**] for plane and 3-D structures. Principal stresses are recalculated from the component stresses and are invariant with coordinate system. Use **PLSECT**command to display stresses.

Stresses through a section may be linearized and separated into categories for various code calculations. Linearized stresses may be printed [**PRSECT**] and displayed [**PLSECT**]. The section is defined by a path consisting of two end points (nodes) and 47 intermediate points (automatically determined by linear interpolation in the active display coordinate system [**DSYS**]), regardless of the number of divisions set by **PATH**,,,,*nDiv*. The values of the component stresses to be linearized are interpolated at the path points within each path element from the element's average corner nodal values. These values will be dependent on the active coordinate system. If **RSYS**, SOLU or **RSYS**, -1 are active, the linearized stresses will be calculated and printed in the global Cartesian coordinate system. The section path may be through any solid (2-D plane, 2-D axisymmetric, or 3-D) elements. Section paths, which may be anywhere, are usually defined to be through the thickness of the structure and normal to the inner and outer structure surfaces. Section paths (in-plane only) may also be defined for shell element structures. See the *ANSYS, Inc. Theory Reference* for details.

Stress components through the section are linearized by a line integral method and separated into constant membrane stresses, bending stresses varying linearly between end points, and peak stresses (defined as the difference between the actual (total) stress and the membrane plus bending combination).

For nonaxisymmetric structures, the bending stresses are calculated such that the neutral axis is at the midpoint of the path. Axisymmetric results include the effects of both the radius of revolution (automatically determined from the node locations) and the in-plane average radius of curvature of the section surfaces (user input).

For axisymmetric cases, ANSYS calculates the linearized bending stress in the through-thickness direction as the difference between the total outer fiber stress and the membrane stress. The calculation method may be conservative for locations with a highly nonlinear variation of stress in the through-thickness direction. For more information, see the discussion of axisymmetric cases (specifically Equation 19–40) in the *ANSYS, Inc. Theory Reference*.

Portions of this command are not supported by PowerGraphics [**/GRAPHICS**,POWER].

Menu Paths

Main Menu>General Postproc>List Results>Linearized Strs
Main Menu>General Postproc>Path Operations>List Linearized
Utility Menu>List>Results>Linearized Stresses

PRSSOL, *Item*, *Comp*

Prints BEAM188 and BEAM189 section results.

POST1: Results

MP ME ST <> <> <> <> <> <> PP ED

Item
Label identifying the item. Valid item labels are listed below. Some items also require a component label.

Comp
Component of the item (if required). Valid component labels are listed below.

Notes

PRSSOL prints the BEAM188 and BEAM189 section nodal and section integration point results. Stresses and strains are printed at section nodes. Plastic strains, plastic work, and creep strains are printed at section integration points. For example, **PRSSOL**,S,COMP prints the X, XZ, and XY components of stresses at section nodes.

To display a listing of the section nodal and integration point coordinates, use the **SLIST** command.

PRSSOL - Valid Item and Component Labels

Item	Comp	Description
	Valid item and component labels for section results are:	
S	COMP	X, XZ, XY stress components.
"	PRIN	S1, S2, S3 principal stresses, SINT stress intensity, and SEQV equivalent stress.
EPTO	COMP	Total strains (EPEL+EPPL+EPTH).
"	PRIN	Total principal strains, strain intensity, and equivalent strain.
EPPL	COMP	Plastic strain components.
"	PRIN	Principal plastic strains, plastic strain intensity, and equivalent plastic strain.
EPCR	COMP	Creep strain components.
"	PRIN	Principal creep strains, creep strain intensity, and equivalent creep strain.
EPTH		Thermal strains.
NL		Nonlinear item plastic work.
BMOM		Bimoments.

Menu Paths

Main Menu>General Postproc>List Results>Section Solution
Utility Menu>List>Results>Section Solution

PRTIME, *TMIN*, *TMAX*

Defines the time range for which data are to be listed.

POST26: Listing

MP ME ST DY <> PR EM <> FL PP ED

TMIN

Minimum time (defaults to the first point stored).

TMAX

Maximum time (defaults to the last point stored).

Command Default

Use the previously defined range [**TIMERANGE**].

Notes

Defines the time (or frequency) range (within the range stored) for which data are to be listed.

Menu Paths

Main Menu>TimeHist Postpro>Settings>List

PRVAR, *NVAR1*, *NVAR2*, *NVAR3*, *NVAR4*, *NVAR5*, *NVAR6*

Lists variables vs. time (or frequency).

POST26: Listing

MP ME ST DY <> PR EM <> FL PP ED

NVAR1, *NVAR2*, *NVAR3*, *NVAR4*, *NVAR5*, *NVAR6*

Variables to be displayed, defined either by the reference number or a unique thirty-two character name. If duplicate names are used the command will print the data for the lowest-numbered variable with that name.

Notes

Lists variables vs. time (or frequency). Up to six variables may be listed across the line. Time column output format can be changed using the **/FORMAT** command arguments *Ftype*, *NWIDTH*, and *DSIGNF*.

Menu Paths

Main Menu>TimeHist Postpro>List Variables

PRVAROPT, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*, *Lab10*
Lists up to ten optimization parameters.

OPTIMIZATION: Display
MP ME ST DY <> PR EM <> FL PP ED

Lab1, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*, *Lab10*
Names of the parameters to be listed.

Notes

Lists up to ten optimization parameters at a time. Parameters are listed vs. the **XVAROPT** parameter (defaults to set number) in the order corresponding to an ascending order of the **XVAROPT** parameter.

Menu Paths

Main Menu>Design Opt>Design Sets>Graphs/Tables

PRVECT, *Item*, *Lab2*, *Lab3*, *LabP*
Prints results as vector magnitude and direction cosines.

POST1: Results
POST1: Element Table
MP ME ST DY <> PR EM <> FL PP ED

Item
Predefined vector item (from PRVECT - Valid Item and Component Labels below) or a label identifying the i-component of a user-defined vector.

Lab2
Label identifying the j-component of a user-defined vector. Must be blank if *Item* is selected from list below.

Lab3
Label identifying the k-component of a user-defined vector. Must be blank if *Item* is selected from list below or for 2-D user defined vector.

LabP
Label assigned to resultant vector for printout labeling (defaults to *Item*).

Notes

Prints various solution results as vector magnitude and direction cosines for the selected nodes and/or elements. For example, **PRVECT**,U prints the displacement magnitude and its direction cosines for all selected nodes. For nodal degree of freedom vector results, direction cosines are with respect to the results coordinate system RSYS. For element results, direction cosines are with respect to the global Cartesian system. Item components may be printed with the **PRNSOL** command. Various results also depend upon the recalculation method and the selected results location [**LAYER**, **SHELL**, **NSEL**, and **ESEL**]. Items may be selected from a set of recognized vector labels (*Item*) or a vector may be defined from up to three scalar labels (*Item*,*Lab2*,*Lab3*). Scalar labels may be user-defined with the **ETABLE** command.

Portions of this command are not supported by PowerGraphics [**/GRAPHICS**,POWER].

PRVECT - Valid Item and Component Labels

Item	Comp	Description
		Valid item labels for nodal degree of freedom vector results are:
U		Structural displacement vector magnitude and direction cosines.
ROT		Structural rotation vector magnitude and direction cosines.
V		Velocity vector magnitude and direction cosines.
A		Magnetic vector potential vector magnitude and direction cosines.
		Valid item labels for element results are:
S		Principal stresses and direction cosines.
EPTO		Principal total strains (EPEL + EPPL + EPCR) and direction cosines.
EPEL		Principal elastic strains and direction cosines.
EPPL		Principal plastic strains and direction cosines.
EPCR		Principal creep strains and direction cosines.
EPTH		Principal thermal strains and direction cosines.
TG		Thermal gradient vector sum and direction cosines.
TF		Thermal flux vector sum and direction cosines.
PG		Pressure gradient vector sum and direction cosines.
EF		Electric field vector sum and direction cosines.
D		Electric flux density vector sum and direction cosines.
H		Magnetic field intensity vector sum and direction cosines. If *Lab2* is blank, then Item is interpreted as one of the predefined labels. Otherwise, Item is interpreted as a user-defined **ET** label and ANSYS will request a non-blank *Lab2*/*Lab3* according to the dimension of the problem.
B		Magnetic flux density vector sum and direction cosines.
FMAG		Magnetic force vector sum and direction cosines.
P		Poynting vector sum and direction cosines.
JS		Source current density vector sum and direction cosines for low-frequency magnetic analyses. Total current density vector sum and direction cosines (sum of conduction and displacement current densities) in low frequency electric analyses.
JT		Total measureable current density vector sum and direction cosines in low-frequency electromagnetic analyses. (Conduction current density vector sum and direction cosines in a low-frequency electric analysis.)
JC		Conduction current density vector sum and direction cosines for elements that support conduction current calculation.

Menu Paths

Main Menu>General Postproc>List Results>Vector Data
Utility Menu>List>Results>Vector Data

PSCR, *Kywrd*, *KEY*

Specifies various PostScript options.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

If *Kywrd* = CMAP, command format is **PSCR**,CMAP,*INDEX*,*IRED*,*IGRN*,*IBLU*.

INDEX
Color map index (0 to 15, 128 to 255).

IRED
Red intensity (0 to 100).

IGRN
Green intensity (0 to 100).

IBLU
Blue intensity (0 to 100).

If *Kywrd* = LWID, command format is **PSCR**,LWID,*KYLWID*.

KYLWID
Line width factor (1 to 99, initially 3).

If *Kywrd* = COLOR, command format is **PSCR**,COLOR,*KEY*.

KEY
Color key:

0
Monochrome.

1
16 colors.

2
256+ colors.

3
Shaded monochrome.

If *Kywrd* = TRANX, command format is **PSCR**,TRANX,*OFFSET*.

OFFSET
X translational offset (initially 592).

If *Kywrd* = TRANY, command format is **PSCR**,TRANY,*OFFSET*.

OFFSET
Y translational offset (initially 25).

If *Kywrd* = ROTATE, command format is **PSCR**,ROTATE,*ANGLE*.

ANGLE
Rotation angle in degrees (initially 90.0).

If *Kywrd* = SCALE, command format is **PSCR**,SCALE,*SCALE*.

SCALE
Scale factor (initially 0.238).

If *Kywrd* = TIFF, command format is **PSCR**,TIFF,*KEY*. Used to add a Tagged Image File Format (TIFF) bitmap preview image to the encapsulated PostScript file.

KEY
Preview image key:

0
Do not include TIFF bitmap preview image.

1
Include wireframe TIFF bitmap preview image.

2
Include shaded TIFF bitmap preview image.

If *Kywrd* = EPSI, command format is **PSCR**,EPSI,*KEY*. Used to add an Encapsulated PostScript Interchange (EPSI) format bitmap preview image to the encapsulated PostScript file.

KEY
Preview image key:

0
Do not include EPSI bitmap preview image.

1
Include wireframe EPSI bitmap preview image.

2
Include shaded EPSI bitmap preview image.

If *Kywrd* = PAPER, command format is **PSCR**,PAPER,*SIZE*,*ORIENT*.

SIZE
Standard paper sizes, A, B, C, D, E, A4, A3, A1, A0.

ORIENT
Paper orientation, either Landscape or Portrait.

If *Kywrd* = HIRES, command format is **PSCR**,HIRES,*KEY*. Used to select the output resolution.

KEY
Output resolution key:

0
High resolution graphics off. The image is exported as a bitmap.

1
High resolution graphics on. The image will be exported in polygon mode (**/TYPE**, , 4 precise hidden) for maximum printer resolution.

Notes

This command is available in both the ANSYS and DISPLAY programs. It is valid for postscript format files chosen in ANSYS with the **/SHOW**,PSCR command, or in DISPLAY with **/SHOWDISP**,POSTSCRIPT.

An output file is generated for each plot. The ANSYS file is named **Jobname*NN*.pscr**. In the DISPLAY program, this file is named **PSCRnn**. This file remains open for a subsequent **/NOERASE** plot, and will be incomplete until the program is closed (**/EXIT**), or until the next file is opened by the next **/ERASE** plot request.

Issuing **PSCR**,STAT will list paper size, orientation and resolution modes.

Menu Paths

Utility Menu>PlotCtrls>Redirect Plots>To PSCR File

PSDCOM, *SIGNIF*, *COMODE*

Specifies the power spectral density mode combination method.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

SIGNIF

Combine only those modes whose significance level exceeds the*SIGNIF* threshold. For PSD response (**SPOPT**,PSD), the significance level is defined as the modal covariance matrix term, divided by the maximum modal covariance matrix term. Any term whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of terms used. *SIGNIF* defaults to 0.0001. If *SIGNIF* is specified as 0.0, it is taken as 0.0.

COMODE

First *COMODE* number of modes to be actually combined. *COMODE* must always be less than or equal to *NMODE* (input quantity *NMODE* on the **SPOPT** command). *COMODE* defaults to *NMODE*. *COMODE* performs a second level of control for the first sequential *COMODE* number of modes to be combined. It uses the significance level threshold indicated by *SIGNIF* and operates only on the significant modes.

Notes

This command is also valid for PREP7. This command is valid only for **SPOPT**,PSD.

Product Restrictions

PSDCOM is not allowed in ANSYS Professional.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Mode Combine

PSDFRQ, *TBLNO1*, *TBLNO2*, *FREQ1*, *FREQ2*, *FREQ3*, *FREQ4*, *FREQ5*, *FREQ6*, *FREQ7*

Defines the frequency points for the input spectrum vs. FREQ tables of PSD and multi-point spectrum analyses.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

TBLN01

Input table number. When used with the **COVAL** or the **QDVAL** command, *TBLN01* represents the row number of this table. Up to 10 tables may be defined.

TBLN02

Input table number. *TBLN02* is used only for the **COVAL** or the **QDVAL** commands and represents the column number of this table.

FREQ1, *FREQ2*, *FREQ3*, *FREQ4*, *FREQ5*, *FREQ6*, *FREQ7*

Frequency points (cycles/time) for spectrum vs. frequency tables. *FREQ1* should be greater than zero, and values must be in ascending order. Log-log interpolation will be used between frequency points.

Notes

The spectrum values may be input with the **PSDVAL** (for **SPOPT**,PSD and **SPOPT**,MPRS), **COVAL** (for **SPOPT**,PSD only), or **QDVAL** (for **SPOPT**,PSD only) commands. A separate **PSDFRQ** command must be used for each table and cross table defined. Frequencies must be in ascending order.

Repeat **PSDFRQ** command for additional frequency points (50 maximum). Values are added after the last nonzero frequency. If all fields after **PSDFRQ** are blank, all input vs. frequency tables are erased. If *TBLN01* is nonblank, all corresponding **PSDVAL** tables are erased. If both *TBLN01* and *TBLN02* are nonblank, all corresponding **COVAL** and **QDVAL** tables are erased.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Erase Tables
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Spect vs Freq
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Cospectral
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Erase Co & Quad
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Erase Tables
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>PSD vs Freq
Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Erase Tables
Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Spect vs Freq
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Cospectral
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Erase Co & Quad
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Erase Tables
Main Menu>Solution>Load Step Opts>Spectrum>PSD>PSD vs Freq

PSDGRAPH, *TBLNO1*, *TBLNO2*

Displays input PSD curves

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

TBLN01
PSD table number to display.

TBLN02
Second PSD table number to display. *TBLN02* is used only in conjunction with the **COVAL** or the **QDVAL** commands.

Notes

The input PSD tables are displayed in log-log format as dotted lines. The best-fit curves, used to perform the closed-form integration, are displayed as solid lines. If there is a significant discrepancy between the two, then you should add one or more intermediate points to the table to obtain a better fit.

If *TBLN02* is zero, blank, or equal to *TBLN01*, then the autospectra (**PSDVAL**) are displayed for *TBLN01*. If *TBLN02* is also specified, then the autospectra for *TBLN01* and *TBLN02* are displayed, along with the corresponding co-spectra (**COVAL**) and quadspectra (**QDVAL**), if they are defined.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Spectrum>Graph PSD Tab
Main Menu>Solution>Spectrum>Graph PSD Tab

PSDRES, *Lab*, *RelKey*

Controls solution output written to the results file from a PSD analysis.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

Lab
Label identifying the solution output:

DISP
Displacement solution (default). One-sigma displacements, stresses, forces, etc. Written as load step 3 on **File.RST**.

VELO
Velocity solution. One-sigma velocities, "stress velocities," "force velocities," etc. Written as load step 4 of **File.RST**.

ACEL
Acceleration solution. One-sigma accelerations, "stress accelerations," "force accelerations," etc. Written as load step 5 on **File.RST**.

RelKey
Key defining relative or absolute calculations:

REL
Calculations are relative to the base excitation (default).

ABS
Calculations are absolute.

OFF
No calculation of solution output identified by *Lab*.

Command Default

Relative displacement solution, no velocity or acceleration solution for 1 σ results.

Notes

Controls the amount and form of solution output written to the results file from a PSD analysis. One-sigma values of the relative or absolute displacement solution, relative or absolute velocity solution, relative or absolute acceleration solution, or any combination may be included on the results file.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Calc Controls
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Calc Controls

PSDSPL, *TBLNO*, *RMIN*, *RMAX*

Defines a partially correlated excitation in a PSD analysis.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

TBLNO
Input PSD table number defined with **PSDVAL** command.

RMIN
Minimum distance between excitation points which are partially correlated. Excited nodes closer than *RMIN* will be fully correlated.

RMAX
Maximum distance between excitation points which are partially correlated. Excited nodes farther apart than *RMAX* will be uncorrelated.

Notes

Defines a partially correlated excitation in terms of a sphere of influence relating excitation point geometry (in a PSD analysis). If the distance between any two excitation points is less than *RMIN*, then the excitation is fully correlated. If the distance is greater than *RMAX*, then the excitation is uncorrelated. If the distance lies between *RMIN* and *RMAX*, then the excitation is partially correlated with the degree of correlation dependent on the separation distance between the points. This command is not available for a pressure PSD analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Spatial Correlat
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Spatial Correlat

---

PSDUNIT, *TBLNO*, *Type*, *GVALUE*

Defines the type of PSD or multi-point response spectrum.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

TBLNO
Input table number.

Type
Label identifying the type of spectrum:

DISP
Displacement spectrum (in terms of displacement^2/Hz for PSD).

VELO
Velocity spectrum (in terms of velocity^2/Hz for PSD).

ACEL
Acceleration spectrum (in terms of acceleration^2/Hz for PSD).

ACCG
Acceleration spectrum (in terms of g^2/Hz for PSD).

FORC
Force spectrum (in terms of force^2/Hz for PSD).

PRES
Pressure spectrum (in terms of pressure^2/Hz for PSD).

GVALUE
Value of acceleration due to gravity in any arbitrary units for ACCG PSD table. Default is 386.4 in/sec^2.

Command Default

Acceleration (ACEL) spectrum (acceleration^2/Hz).

Notes

Defines the type of PSD or multi-point response spectrum defined by the **PSDVAL**, **COVAL**, and **QDVAL** commands.

Force (FORC) and pressure (PRES) type spectra can be used only as a nodal excitation.

GVALUE is valid only when type ACCG is specified. A zero or negative value cannot be used. A parameter substitution can also be performed.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Settings
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Settings
Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Settings
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Settings

PSDVAL, *TBLNO*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*

Defines PSD or multi-point response spectrum values.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

TBLN0
Input table number being defined.

SV1, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*
Spectral values corresponding to the frequency points [**PSDFRQ**]. Values are interpreted as defined with the **PSDUNIT** command.

Notes

Defines PSD or multi-point response spectrum values to be associated with the previously defined frequency points. Repeat **PSDVAL** command for additional values, up to the number of frequency points [**PSDFRQ**]. Values are added after the last nonzero frequency.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Spect vs Freq
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>PSD vs Freq
Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Spect vs Freq
Main Menu>Solution>Load Step Opts>Spectrum>PSD>PSD vs Freq

PSDWAV, *TBLNO*, *VX*, *VY*, *VZ*

Defines a wave propagation excitation in a PSD analysis.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

TBLN0
Input PSD table number defined with **PSDVAL** command.

VX
Global Cartesian X-velocity of traveling wave.

VY
Global Cartesian Y-velocity of traveling wave.

VZ
Global Cartesian Z-velocity of traveling wave.

Notes

Defines a traveling wave in a PSD analysis. This command is not available for a pressure PSD analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Traveling Wave
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Traveling Wave

---

PSEL, *Type*, *Pname1*, *Pname2*, *Pname3*, *Pname4*, *Pname5*, *Pname6*, *Pname7*, *Pname8*, *Pname9*, *Pname10*
Selects a path or paths.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of select:

S
Select a new path.

R
Reselect a path from the current set of paths.

A
Additionally select a path and extend the current set of paths.

U
Unselect a path from the current set of paths.

ALL
Restore the full set of paths.

NONE
Unselect the full set of paths.

INV
Invert the current set of paths (selected becomes unselected and vice versa).

Pname1, *Pname2*, *Pname3*, *Pname4*, *Pname5*, *Pname6*, *Pname7*, *Pname8*, *Pname9*, *Pname10*
Name of existing path(s).

Notes

Selects a path or multiple paths, up to ten. Data are flagged as selected and unselected; no data are actually deleted from the database. There is no default for this command; you must specify a type and pathname.

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Store>Paths in file
Main Menu>Preprocessor>Path Operations>Store>Paths in file

/PSF, *Item*, *Comp*, *KEY*, *KSHELL*, *Color*

Shows surface load symbols on model displays.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

Item, *Comp*
Labels identifying the surface load to be shown; see /PSF - Valid Item and Component Labels.

KEY
Key to turn surface load symbols on or off:

0
Off (default).

1
On, shown as face outlines. Line surface loads [**SFL**] on solid model plots are shown as arrows.

2
On, shown as arrows.

3
On, shown as color filled surfaces. Line and area surface loads [**SFL** and **SFA**] on solid model plots are shown as arrows.

KSHELL
Visibility key for shell elements.

0
Off (default), surface load symbols are displayed only on visible load faces.

1
On, surface load symbols are displayed even if load face is not visible.

Color
Visibility key for contour legend.

ON
The symbols (arrows or face outlines) will show up in color with the legend showing the corresponding color labels (default).

OFF
The contour legend will not be displayed. The symbols (arrows or face outlines) will show up in grey. The size of the arrows will be proportional to the applied load.

Command Default

No surface load symbols are displayed.

Notes

This command determines whether and how to show surface loads on subsequent model displays. If surface loads are applied to solid model entities, only solid model plots will show the load symbols; node and element plots will not show them unless the loads are transferred [**SFTRAN** or **SBCTRAN**]. Similarly, if surface loads are applied to nodes and elements, solid model plots will not show the load symbols. For node and element plots of shell element models, the surface load symbols will be shown only if the load face is visible from the current viewing direction.

The effects of the **/PSF** command are not cumulative (that is, the command does not modify an existing setting from a previously issued **/PSF** command). If you issue multiple **/PSF** commands during an analysis, only the setting specified by the most recent **/PSF** command applies.

When you use the Radiosity method (*Item* = RDSF and *Comp* = ENCL) and you set *Key* = 2, the radiation arrows point outward from any element face.

Issue **/PSF**,STAT to display current **/PSF** settings, and **/PSF**,DEFA to reset them back to default. Other useful commands are **/PNUM**,SVAL,1 to show the values of the surface loads, **/VSCALE** to change arrow lengths, and **/PBC** and **/PBF** to turn on other load symbols.

For BEAM elements, only the colors representing shear (GREEN) and normal (RED) pressures are displayed for the arrows. The color of these arrows does not correspond to the magnitudes in the contour legend. The length of these arrows does, however, correlate to the relative magnitude of the pressures.

This command is valid in any processor.

/PSF - Valid Item and Component Labels

Item	Comp	Description
		Valid item and component labels are:
PRES	NORM	Applied structural pressure normal to the face (real component).
"	TANX	Applied structural pressure in the tangential-x direction (real component).
"	TANY	Applied structural pressure in the tangential-y direction (real component).
“	INRM	Applied structural pressure normal to the face (imaginary component).
“	ITNX	Applied structural pressure in the tangential-x direction (imaginary component).
“	ITNY	Applied structural pressure in the tangential-y direction (imaginary component).
CONV	HCOEF	Applied convection (film coefficient).
"	TBULK	Applied convection (bulk temperature).
RAD	EMIS	Applied radiation (emissivity).
"	TAMB	Applied radiation (ambient temperature).
RDSF	EMSS	Radiation emissivity.
“	ENCL	Enclosure number.
FSIN		Fluid-solid interface number.
VFRC		Volume fraction (VOF method).

Item	Comp	Description
HFLUX		Applied heat flux.
FSI		Acoustic fluid-structure interface flag.
IMPD		Applied acoustic impedance.
SHLD	COND	Applied conductivity.
"	MUR	Applied relative permeability.
MXWF		Maxwell force flag.
INF		Exterior surface flag.
CHRGS		Applied electric surface charge density.
MCI		Magnetic circuit interface.

Menu Paths

Utility Menu>PlotCtrls>Symbols

PSMESH, *SECID*, *Name*, *P0*, *Egroup*, *NUM*, *KCN* ,*KDIR*, *VALUE*, *NDPLANE*, *PSTOL*, *PSTYPE*, *ECOMP*, *NCOMP*

Create and mesh a pretension section

PREP7: Meshing

MP ME ST <> <> <> <> <> <> PP ED

SECID

Unique section number. This number must not already be assigned to a section.

Name

Unique eight character descriptive name, if desired.

P0

Pretension node number. The node will be defined if it doesn't exist and the number defaults to the highest node number plus one.

Egroup, *NUM*

Element group on which **PSMESH** will operate. If *Egroup* = P, graphical picking is enabled and *NUM* is ignored (valid only in the GUI).

L (or LINE)

PSMESH operates on all elements in the line specified by *NUM*. New pretension nodes are associated with *NUM* or entities below it. Any subsequent **LCLEAR** operation of *NUM* deletes the pretension elements and nodes created by **PSMESH**.

A (or AREA)

PSMESH operates on all elements in the area specified by *NUM*. New pretension nodes are associated with *NUM* or entities below it. Any subsequent **ACLEAR** of *NUM* deletes the pretension elements and nodes created by **PSMESH**.

V (or VOLU)

PSMESH operates on all elements in the volume specified by *NUM*. New pretension nodes are associated with *NUM* or entities below it. Any subsequent **VCLEAR** of *NUM* deletes the pretension elements and nodes created by **PSMESH**.

P

PSMESH operates on elements selected through the subsequent picking operations, and *NUM* is ignored

ALL
The command operates on all selected elements, and *NUM* is ignored.

KCN
Coordinate system number for the separation surface and normal direction.

KDIR
Direction (x, y, or z) normal to separation surface in the *KCN* coordinate system.

If *KCN* is cartesian, the pretension section normal will be parallel to the *KDIR* axis regardless of the position of the pretension node.

If *KCN* is non-cartesian, the pretension section normal will be aligned with the *KDR* direction of system *KCN* at the position of the pretension node.

VALUE
Point along the *KDIR* axis at which to locate the separation surface. Ignored if *NDPLANE* is supplied.

NDPLANE
Existing node that **PSMESH** will use to locate the separation surface. If *NDPLANE* is supplied, the location of the separation surface is defined by the *KDIR* coordinate of *NDPLANE*.

PSTOL
Optional absolute tolerance about *VALUE*. Allows nodes occurring slightly above or below the separation to be grouped properly. The following expression represents the default value:

$$\sqrt{\frac{\Delta X^2 + \Delta Y^2 + \Delta Z^2}{1000}}$$

where ΔX, ΔY, and ΔZ are the dimensions of the model based on nodal locations (that is, $\Delta X = X_{max} - X_{min}$).

PSTYPE
If specified, this value is the type number for pretension elements. (If not specified, ANSYS defines this value.) If already defined, it must be of type PRETS179.

ECOMP
If specified, the name of a component to be composed of new pretension elements and existing elements modified by the **PSMESH** command.

NCOMP
Name of a component to be composed of nodes on new pretension elements.

Notes

The **PSMESH** command creates a pretension section normal to the pretension load direction by cutting the mesh along existing element boundaries at the point defined by *VALUE* or *NDPLANE* and inserting PRETS179 elements. The **PSMESH** command verifies that *PSTYPE* is PRETS179; if it is not, the command finds the lowest available *ITYPE* that is PRETS179, or if necessary will create a new one.

When it is necessary to define the pretension node, ANSYS uses node *NDPLANE*. If the *NDPLANE* value is not specified, ANSYS defines the pretension node at:

- The centroid of geometric entity *NUM*, if *Egroup* = LINE, AREA or VOLU
- The centroid location of all selected elements, if *Egroup* = ALL or if graphical picking is used.

If the elements to which the pretension load is to be applied have already been meshed in two groups, **PSMESH** cannot be used to insert the pretension elements. The **EINTF** command must be used to insert the PRETS179 elements between the two meshed groups.

The **PSMESH** operation copies any nodal temperatures you have defined on the split surface of the original mesh from the original nodes to the newly created coincident duplicate nodes. However, displacements, forces, and other boundary conditions are not copied.

By mathematical definition, the pretension surface must always be a flat plane. In a non-Cartesian coordinate system, the **PSMESH** command creates that plane at the indicated position, oriented with respect to the specified direction of the active system (in the same manner that the **NROTAT** command orients a nodal system with respect to a curved system). For example, assuming a X = 1 and Y = 45 in a cylindrical coordinate system with Z as the axis of rotation (*KCN* = 1), a pretension surface normal to X tilts 45 degrees away from the global X axis.

The **PSMESH** command is valid for structural analyses only.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Elements in Area
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Elements in Line
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Elements in Volu
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Picked Elements
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Selected Element
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Area
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Line
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Volu
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Picked Elements
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Selected Element
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Area
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Line
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Volu
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Picked Elements
Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Selected Element
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Elements in Area
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Elements in Line
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Elements in Volu
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Picked Elements
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Selected Element
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Area

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Line
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Volu
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Picked Elements
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Selected Element
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Area
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Line
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Volu
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Picked Elements
Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Selected Element

---

PSOLVE, *Lab*

Directs the program to perform a partial solution.

SOLUTION: Analysis Options

MP ME ST <> <> <> <> <> <> <> ED

Lab

Valid labels defining the solution step. All characters are required:

CGSOL
: Calculates the DOF solution using the Jacobi conjugate gradient solver. Requires **File.FULL**. Displacements are updated.

EIGDAMP
: Calculates the eigenvalues and eigenvectors using the damped eigensolver. Requires **File.FULL** from **MODOPT**,UNSYM or **MODOPT**,DAMP options. Produces **File.MODE**.

EIGEXP
: Expands the eigenvector solution. Requires **File.MODE**. Produces **File.RST**.

EIGFULL
: Calculates the eigenvalues and eigenvectors using full subspace. Requires **File.FULL** from **MODOPT**,SUBSP option. Produces **File.MODE**.

EIGLANB
: Calculates the eigenvalues and eigenvectors using Block Lanczos. Requires **File.FULL** from **MODOPT**,LANB option. Produces **File.MODE**.

EIGREDUC
: Calculates the eigenvalues and eigenvectors using Householder. Requires **File.REDM**. Produces **File.MODE**.

EIGUNSYM
: Calculates the eigenvalues and eigenvectors using the unsymmetric eigensolver. Requires **File.FULL** from **MODOPT**,UNSYM or **MODOPT**,DAMP options. Produces **File.MODE**.

ELFORM
Creates the element matrices. Produces **File.EMAT** and **File.ESAV**.

Note — If you want to include prestress effects (**PSTRES**,ON) from a previous prestress analysis, the ELFORM option requires the **File.EMAT** and **File.ESAV** files generated by that analysis.

ELPREP
Modifies element matrices for solution and calculates inertia relief terms (**IRLF**). Requires **File.EMAT**. Produces **File.EROT**.

REDWRITE
Writes the reduced matrix to a file. Requires **File.REDM**. Produces **File.SUB**.

TRIANG
Triangularizes the matrices completely (or to the master degrees of freedom, if appropriate) or assembles the global matrices depending on the analysis type and options. Requires **File.EMAT**. Produces **File.TRI** (and **File.REDM**) or **File.FULL**.

Notes

Directs the program to perform a partial solution (that is, one step of an analysis sequence). Predefined analysis types (**ANTYPE**) perform a defined subset of these solution steps in a predefined sequence. You can use the partial solution procedure to repeat a certain step of an analysis or to restart an analysis.

Not all steps are valid for all analysis types. The order of the steps may vary depending on the result you desire. See the *ANSYS Basic Analysis Guide* for a description of how to perform partial and predefined solutions.

Issue a **PSOLVE** series of commands in a single load step. The **SOLVE** command (which executes the **PSOLVE** load step) should appear in a separate load step, as shown in this example:

```
/solu
psolve,elform
psolve,elprep
psolve,triang
fini
/solu
kuse,1
solve
fini
```

In a cyclic symmetry analysis, **PSOLVE**,EIGLANB performs the modal analysis at multiple load steps, one for each nodal-diameter specified via the **CYCOPT** command. In addition, the eigenvector solution is expanded at each nodal-diameter solution, eliminating the need for a separate expansion pass (**PSOLVE**,EIGEXP).

If issuing **PSOLVE**,ELFORM and **PSOLVE**,ELPREP using the Jacobi Conjugate Gradient solver, do so only *after* issuing **PSOLVE**,CGSOL; otherwise, unpredictable results may occur.

Although documented to work, using the **PSOLVE** commands with an iterative solver is not likely to decrease solution-processing time.

If **File.EMAT** is required (ELFORM with **PSTRES**,ON; ELPREP; or TRIANG), run the prior analysis with **EMATWRITE**,YES to ensure that a **File.EMAT** is generated.

In a prestressed modal analysis, issue a **PSOLVE**,TRIANG command immediately before a **PSOLVE**,EIGSYM, **PSOLVE**,EIGUNSY, **PSOLVE**,EIGDAMP or **PSOLVE**,EIGREDUC command to ensure that ANSYS creates a proper

.FULL file. (The **PSOLVE**,EIGFULL and **PSOLVE**,EIGLANB commands do not require a preceding **PSOLVE**,TRIANG command but, if you issue it anyhow, the **PSOLVE**,EIG*xxxx* commands will still work correctly.)

Menu Paths

Main Menu>Solution>Solve>Partial Solu

PSPEC, *MAT*, *DNOM*, *SCHED*, *OD*, *TK*

Defines pipe material and dimensions.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

MAT

Material number referring to a material property [**MP**]. Material number must be between 1 and 40.

DNOM, *SCHED*

Nominal diameter of pipe and schedule rating. Only valid ratings accepted. If these are specified, the *OD* and *TK* values are found from an internal table.

Valid values for *DNOM* are: 1, 1.5, 2, 2.5, 3, 3.5, 4, 5, 6, 8, 10, 12, 14, 16, 18, 20, 22, 24, 26, 28, 30, 32, 34, and 36.

Valid ratings for *SCHED* are: 5, 5S, 10, 10S, 20, 30, 40, 40S, 60, 80 80S, 100, 120, 140, 160, XS, XXS, and STD.

OD

Outer diameter of pipe (if *DNOM* not specified). If both *DNOM* and *OD* are not specified, *OD* and *TK* retain their previous values.

TK

Wall thickness of pipe (if *OD* specified).

Notes

Defines pipe material and dimensions. See the PREP7 **RUN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Specifications

---

/PSPEC, *PCOLOR*, *KFILL*, *KBORDR*

Creates annotation polygon attributes (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

PCOLOR

Polygon color ($0 \leq$ *PCOLOR* ≤ 15):

0

Black.

1
Red-Magenta.

2
Magenta.

3
Blue-Magenta.

4
Blue.

5
Cyan-Blue.

6
Cyan.

7
Green-Cyan.

8
Green.

9
Yellow-Green.

10
Yellow.

11
Orange.

12
Red.

13
Dark Gray.

14
Light Gray.

15
White.

KFILL
Polygon fill key:

0
Hollow polygon.

1
Filled polygon.

KBORDR
Polygon border key:

0
No border.

1
Border.

Notes

Creates annotation polygon attributes to control certain characteristics of the polygons created via the **/POLYGON**, **/PMORE**, **/PCIRCLE** and **/PWEDGE** commands. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

PSPRNG, *NLOC*, *TYPE*, *K*, *DX*, *DY*, *DZ*, *ELEM*

Defines a spring constraint in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

NLOC
Node where spring is to be placed. Defaults to current run starting point.

TYPE
Type of spring:

TRAN
Translational (default).

ROT
Rotational.

K
Spring constant value (must be positive).

DX, *DY*, *DZ*
Increment (in terms of the active coordinate system components) to determine spring ground point. Spring length must not be zero. Constraints are automatically generated at the ground point.

ELEM
Element number to be assigned to spring (defaults to the previous maximum element number (MAXEL + 1)).

Notes

Defines a spring constraint (spring element COMBIN14) at a given location in a piping run. See the PREP7 **RUN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Spring Support

---

/PSTATUS, *WN*

Displays the global or window display specifications.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number for status (defaults to global specifications).

Notes

Displays the current global or window display specifications. Global display specifications are common to all windows (e.g. **/SHOW**, etc.). Window display specifications are specific to one window (e.g. **/VIEW**, **/TYPE**, etc.).

This command is valid in any processor.

Menu Paths

Utility Menu>List>Status>Graphics>General
Utility Menu>List>Status>Graphics>Window 1
Utility Menu>List>Status>Graphics>Window 2
Utility Menu>List>Status>Graphics>Window 3
Utility Menu>List>Status>Graphics>Window 4
Utility Menu>List>Status>Graphics>Window 5

PSTRES, *Key*

Specifies whether prestress effects are calculated or included.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR <> <> <> PP ED

Key
Prestress key:

OFF
Do not calculate (or include) prestress effects (default).

ON
Calculate (or include) prestress effects.

Notes

Specifies whether or not prestress effects are to be calculated or included. Prestress effects are calculated in a static or transient analysis for inclusion in a buckling, modal, harmonic (Method = FULL or REDUC), transient (Method = REDUC), or substructure generation analysis. If used in SOLUTION, this command is valid only within the first load step.

If the prestress effects are to be calculated in a nonlinear static or transient analysis (for a prestressed modal analysis of a large-deflection solution), you can issue a **SSTIF**,ON command (rather than a **PSTRES**,ON command) in the static analysis.

If thermal body forces are used during a static analysis for the calculation of prestress effects, these thermal body forces should not be deleted during any subsequent full harmonic response analyses. If these body forces are deleted, the thermal prestress effects will not be included in the harmonic response analysis. Temperature loads used to define the thermal prestress will also be used in the full harmonic response analysis as sinusoidally time-varying temperature loads.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic
Main Menu>Solution>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Sol'n Controls>Basic

/PSYMB, *Label*, *KEY*

Shows various symbols on displays.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

Label

Show symbols as selected from the following labels:

CS
Local coordinate systems.

NDIR
Nodal coordinate systems (on rotated nodes only).

ESYS
Element coordinate systems (element displays only).

LDIR
Line directions (line displays only).

LDIV
Controls the display of element divisions on lines.

ADIR
Area direction symbol (for keypoint, line, area and volume plots).

LAYR
Layer orientations (relative to the projected element x-axis). Used only with layered elements in an element display. Use *KEY* for layer number.

PCON
Convergence criterion symbols on nodes (for a p-method analysis).

ECON
Element mesh symbols on keypoints and lines.

DOT
Larger symbols displayed for node and keypoint locations. When *Label* = DOT, *KEY* = 1 by default.

XNOD
Extra node of surface or circuit elements.

FBCS
Force boundary condition scaling. Subsequent *KEY* value determines whether or not to scale the applied and derived forces/moments to the same maximum value.

DEFA
Resets the symbol keys so that ANSYS displays none of the symbols controlled by the **/PSYMB** command. The value of the *KEY* field is ignored.

STAT
Prints the status of the settings of the symbol keys controlled by the **/PSYMB** command. The *KEY* field is ignored.

KEY
Symbol key:

0
No symbol (default). If *Label* = LDIV, then *KEY*= 0 indicates that the displayed element divisions will correspond to the existing mesh (the word MESHED or EXISTING can also be substituted). Also, for *Label* = LDIV, if you execute any meshing command (such as **AMESH** or **VMESH**), *KEY* is set to 0 (MESHED) automatically. If *Label* = FBCS, then *KEY*= 0 indicates that boundary condition scaling will not be common. The applied and derived forces/moments will be scaled to their respective maximum values.

1
Include symbol. If *Label* = LDIV, then *KEY* = 1 indicates that the displayed line divisions will correspond to the value assigned by **LESIZE** (the word LESIZE can also be substituted). Also, for *Label* = LDIV, if you execute the **LESIZE** command, *KEY* is set to 1 (LESIZE) automatically. If *Label* = FBCS, then *KEY*= 1 indicates that boundary condition scaling will be common. The applied and derived forces/moments will be scaled to the same maximum value.

N
If *Label* = LAYR, then *N* is equal to the layer number. If *Label* = DOT, then *N* can be equal to 0,1,.....15, indicating the dot size. If *Label* = LDIV, then *KEY* = -1, indicates that no element divisions will be displayed (the word OFF can also be substituted).

Notes

Includes various symbols on the display. Triads are right-handed with x displayed as the longest leg. Where color is displayed, x is white, y is green, and z is blue. For beams, x is always along the length of the element. For lines, an arrow represents the direction of a line, from the beginning keypoint to the end keypoint. See **/PLOPTS** command for additional display options. Use **/PSTATUS** or **/PSYMB**,STAT to display settings. Use **/PSYMB**,DEFA to reset all specifications back to their defaults. The command **/PSYMB**,ECON,1 causes the symbol "M" to be displayed on keypoints and lines associated with meshed entities. When you issue the command **/PSYMB**,DOT,1, a larger symbol is displayed for each node and keypoint location.

PowerGraphics [**/GRAPHICS**,POWER] does not support **/PSYMB**,ESYS and **/PSYMB**,LAYR.

If *KEY* = *N* and PowerGraphics is off, the centroid of the surface elements is connected to the extra node using a gray line. However, if PowerGraphics is on, the color of the line connecting the centroid to the extra node is the same as that for the elements themselves (as determined by **/PNUM**).

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Areas
Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Lines
Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>with Area
Main Menu>Preprocessor>Trefftz Domain>TZ Symmetry
Utility Menu>PlotCtrls>Symbols

PTEMP, *TOUT*, *TIN*

Defines the pipe wall temperatures in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

TOUT
Outer pipe wall temperature. If NONE, reset temperature specification to none (**BFUNIF** will be assigned).

TIN
Inner pipe wall temperature (defaults to *TOUT*).

Command Default

Assign uniform temperature **BFUNIF** to elements.

Notes

Defines the pipe wall temperatures in a piping run. These temperatures are assigned to the elements as they are generated. See the PREP7 **RUN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Loads

PTXY, *X1*, *Y1*, *X2*, *Y2*, *X3*, *Y3*, *X4*, *Y4*

Defines coordinate pairs for use in polygons and prisms.

PREP7: Primitives

MP ME ST DY <> PR EM <> FL PP ED

X1, *Y1*, *X2*, *Y2*, *X3*, *Y3*, *X4*, *Y4*
X and Y coordinate pairs on the working plane.

Notes

Defines coordinate pairs for use in polygons and prisms [**POLY**, **RPRISM**]. The coordinates must be in the Cartesian coordinate system. The coordinate pairs must be input in a continuous order. **PTXY** may be repeated (up to 100 pairs) until the required pairs have been defined. The pairs will be saved until either the **POLY** or **PRISM** command

is entered. Use **PTXY**,STAT to list the saved coordinate pairs. Use **PTXY**,DELE to delete all the saved coordinate pairs. See the **RPOLY**, **RPRISM**, and **RPR4** commands for other ways to create polygons and prisms.

Menu Paths

This command cannot be accessed from a menu.

PUNIT, *KOPT*

Selects the system of length units to be used in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

K0PT

Units key:

0
: Input units are consistent (no conversions are done) (default).

FTIN or 1
: English units (feet A, inch B, fraction of inch C/D). Use A+B+C/D format for **PDRAG**, **BRANCH**, **RUN**, **BEND**, **MITER**, **REDUCE**, **VALVE**, **BELLOW**, **FLANGE**, **PSPRNG**, and **PGAP** commands. Precede by "-" sign for negative coordinates. (Example: 5+6+7/16 for 5 ft. 6-7/16 in., +3 for 3 in., -0+3 for -3 in., +0+9/16 for 9/16 in.).

  The two signs should not be consecutive. A, B, C, and D must be integers. Use B+C/D format for **PSPEC**, **PINSUL**, and **PCORRO** commands. (Example: 2 for 2 in., 3+1/2 for 3-1/2 in., +3/8 for 3/8 in.)

METRIC or 2
: Metric units (meter A, centimeter B, fraction of cm C/D). Use as explained for English units. (Example: 5+6+7/10 for 5 m 6-7/10 cm with **PDRAG** command.)

Command Default

Input units are consistent (no conversions are done).

Notes

Selects the system of length units to be used for the piping commands. Mixed length units require a + sign to delimit (or position) the units in the system and are converted to the smallest unit of the system (inches or centimeters) upon input.

Note — This conversion is local only to pure length units of the piping commands listed. Other units and units for other commands must be input to be consistent with the smallest length unit of the system used.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Specifications

---

PVECT, *Oper*, *LabXR*, *LabYR*, *LabZR*

Interpolates a set of items onto a path.

POST1: Path Operations

MP ME ST DY <> PR EM <> FL PP ED

Oper

Valid operations for geometry operations along a path are:

NORM

Defines a unit normal vector at each interpolation point in the direction of the cross product of the tangent to the path and the active Z axis. Resulting vector components are in the active coordinate system (which must be Cartesian).

TANG

Defines a unit vector tangent to the path at each interpolation point. Vector components are in the active coordinate system (which must be Cartesian).

RADI

Defines the position vector of each interpolation point of the path from the center of the active coordinate system (which must be Cartesian).

LabXR

Label (8 characters maximum) assigned to X-component of the resulting vector.

LabYR

Label (8 characters maximum) assigned to Y-component of the resulting vector.

LabZR

Label (8 characters maximum) assigned to Z-component of the resulting vector.

Notes

Defines and interpolates a set of labeled path items along predefined path [**PATH**] and performs various geometric operations on these path items. A path item must be defined before it can be used with other path operations. Additional path items may be defined with the **PDEF**, **PCALC**, **PDOT**, and **PCROSS** commands. Path items may be listed or displayed with the **PLPATH**, **PLPAGM** and **PRPATH** commands. Path geometry items (XG, YG, ZG, S) are automatically interpolated (in the active CSYS) if not done so previously with the **PDEF** command.

Menu Paths

Main Menu>General Postproc>Path Operations>Unit Vector

/PWEDGE, *XCENTR*, *YCENTR*, *XLRAD*, *ANGLE1*, *ANGLE2*

Creates an annotation wedge (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

XCENTR

Wedge X center location (-1.0 < X < 2.0).

YCENTR

Wedge Y center location (-1.0 < Y < 1.0).

XLRAD
Wedge radius length.

ANGLE1
Starting angle of wedge.

ANGLE2
Ending angle of wedge. The wedge is drawn counterclockwise from the starting angle, *ANGLE1*, to the ending angle, *ANGLE2*.

Notes

Creates an annotation wedge to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All wedges are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC** and the **/PSPEC** command to set the attributes of the wedge.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

Q Commands

QDVAL, *TBLNO1*, *TBLNO2*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*

Defines PSD quadspectral values.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

TBLN01
First input PSD table number associated with this spectrum.

TBLN02
Second input PSD table number associated with this spectrum.

SV1, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*
PSD quadspectral values corresponding to the frequency points [**PSDFRQ**].

Notes

Defines PSD quadspectral values to be associated with the previously defined frequency points. Repeat **QDVAL** command with the same table number for additional points (50 maximum per curve). Unlike autospectra [**PSDVAL**], the quadspectra can be positive or negative. The quadspectral curve segment where there is a sign change is interpolated linearly (the rest of the curve segments use log-log interpolation). For better accuracy, choose as small a curve segment as possible wherever a sign change occurs.

Two table numbers are required since values are off-diagonal terms. This command is valid for **SPOPT**,PSD only.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral

---

QFACT

Calculates the quality factor for high-frequency electromagnetic resonators.

POST1: Magnetics Calculations

MP ME <> <> <> <> EM <> <> PP ED

Notes

The **QFACT** command macro calculates the quality factor for high-frequency electromagnetic resonators. It returns the quality factor as a scalar parameter, **QFACT**. To compute the quality factor, the macro uses the stored energy, surface losses, and dielectric losses.

See magnetic macros for further details.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Cavity>Q-factor

QSOPT, *Opt*

Specifies quasi static radiation options.

SOLUTION: Radiosity

MP ME <> <> <> PR <> <> <> PP ED

Opt

Quasi static option:

OFF
: Do not run transient radiation problem to steady-state (default).

ON
: Run transient radiation problem to steady-state.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Quasi-Static
Main Menu>Solution>Load Step Opts>Time/Frequenc>Quasi-Static

QUAD, *NODE1*, *NINTR*, *NODE2*, *NFILL*, *NSTRT*, *NINC*, *PKFAC*

Generates a quadratic line of nodes from three nodes.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

NODE1

Begin fill-in from this node location. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NINTR

Intermediate or guiding node. Quadratic curve will pass through this location. *NINTR* may have any node number and any location. If the quadratic line also generates a node with number *NINTR*, the generated location overrides the previous *NINTR* location.

NODE2

End quadratic fill-in at this node location.

NFILL

Fill-in *NFILL* nodes between *NODE1* and *NODE2* (defaults to |*NODE2*-*NODE1*|-1). *NFILL* must be positive.

NSTRT

Node number assigned to first filled-in node (defaults to *NODE1* + *NINC*).

NINC

Add this increment to each of the remaining filled-in node numbers (may be positive or negative). Defaults to (*NODE2*-*NODE1*)/(*NFILL* + 1), i.e., linear interpolation.

PKFAC

Peak location factor. If *PKFAC*=0.5, the peak of the quadratic shape occurs at the *NINTR* location. If 0.0 < *PKFAC* < 0.5, the peak occurs to the *NODE2* side of the *NINTR* location. If 0.5 < *PKFAC* < 1.0, the peak occurs to the *NODE1* side of the *NINTR* location. Defaults to 0.5.

Notes

Generates a quadratic line of nodes (in the active coordinate system) from three nodes. The three nodes determine the plane of the curve and may have been defined in any coordinate system. Any number of nodes may be filled-in and any node number sequence may be assigned.

The quadratic line feature uses three nodes (*NODE1*,*NINTR*,*NODE2*) to determine the plane of the curve. The curve passes through the three points, beginning from *NODE1*, through the intermediate (or guiding) point *NINTR*, and toward *NODE2*.

Generated nodes are also quadratically spaced. If the guiding node number is within the set being generated, it will be relocated according to the quadratic spacing.

The peak location factor is used to determine how the quadratic fits through the three points. Various nodal progressions can be obtained by different combinations of *PKFAC* and the guiding node location. If the guiding node is at mid-length between *NODE1* and *NODE2*, $0.293 \leq PKFAC < 0.707$ will ensure that all generated nodes fall within the *NODE1*,*NODE2* bounds. In the limit, as *PKFAC* approaches 0.0, the peak approaches the line through *NODE1* and *NINTR* at an infinite distance from *NODE1*. The **QUAD** command generates quadratic lines of nodes, which in turn may be used as a base line for generating irregular surfaces of nodes (by repeating [***REPEAT**], generating [**NGEN**, **NSCALE**], etc.). Irregular surfaces may also be generated with the meshing commands.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>Quadratic Fill

/QUIT

Exits a processor.

SESSION: Processor Entry

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is an alternative to the **FINISH** command. If any cleanup or file writing is normally done by the **FINISH** command, it is bypassed if the **/QUIT** command is used instead. A new processor may be entered after this command. See the **/EXIT** command to terminate the run.

This command is valid in any processor. This command is not valid at the Begin level.

Menu Paths

This command cannot be accessed from a menu.

QUOT, *IR*, *IA*, *IB*, --, *Name*, --, --, *FACTA*, *FACTB*

Divides two variables.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA, *IB*
Reference numbers of the two variables to be operated on.

--
Unused field.

Name
Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --
Unused fields.

FACTA, *FACTB*
Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

Notes

Divides two variables according to the operation:

$$IR = (FACTA \times IA)/(FACTB \times IB)$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Divide

R Commands

R, *NSET*, *R1*, *R2*, *R3*, *R4*, *R5*, *R6*
Defines the element real constants.

PREP7: Real Constants

MP ME ST DY <> PR EM <> FL PP ED

NSET
Set identification number (arbitrary). If same as a previous set number, set is redefined. Set number relates to that defined with the element [**REAL**]. Note that the GUI automatically sets this value.

R1, *R2*, *R3*, *R4*, *R5*, *R6*
Real constant values (interpreted as area, moment of inertia, thickness, etc., as required for the particular element type using this set), or table names for tabular input of boundary conditions. Use **RMORE** command if more than six real constants per set are to be input.

Notes

Defines the element real constants. The real constants required for an element are shown in Table 4.*n*.1 of each element description in the *ANSYS Elements Reference*. Constants must be input in the same order as shown in that table. If more than the required number of element real constants are specified in a set, only those required are used. If fewer than the required number are specified, zero values are assumed for the unspecified constants.

If using table inputs (SURF151, SURF152, FLUID116, CONTA171, CONTA172, CONTA173, CONTA174, and CONTA175 only), enclose the table name in % signs (e.g., `%tabname%`).

When copying real constants to new sets, ANSYS recommends that you use the command input. If you do use the GUI, restrict the real constant copy to only the first six real constants (real constants seven and greater will be incorrect for both the master and copy set).

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Real Constants>Add/Edit/Delete
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Mass
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Rotary
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Trans
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Transducer>ElecMech
Main Menu>Preprocessor>Modeling>Create>Circuit>Edit Real Cnst
Main Menu>Preprocessor>Real Constants>Add/Edit/Delete
Main Menu>Solution>Load Step Opts>Other>Real Constants>Add/Edit/Delete

RACE, *XC*, *YC*, *RAD*, *TCUR*, *DY*, *DZ*, --, --, *Cname*

Defines a "racetrack" current source.

PREP7: Special Purpose

MP ME ST DY <> <> EM <> <> PP ED

XC
Location of the mid-thickness of the vertical leg along the working plane X-axis.

YC
Location of the mid-thickness of the horizontal leg along the working plane Y-axis.

RAD
Radius of curvature of the mid-thickness of the curves in the racetrack source. Defaults to .501 * DY

TCUR
Total current, amp-turns (MKS), flowing in the source.

DY
In-plane thickness of the racetrack source.

DZ
Out-of-plane thickness (depth) of the racetrack source.

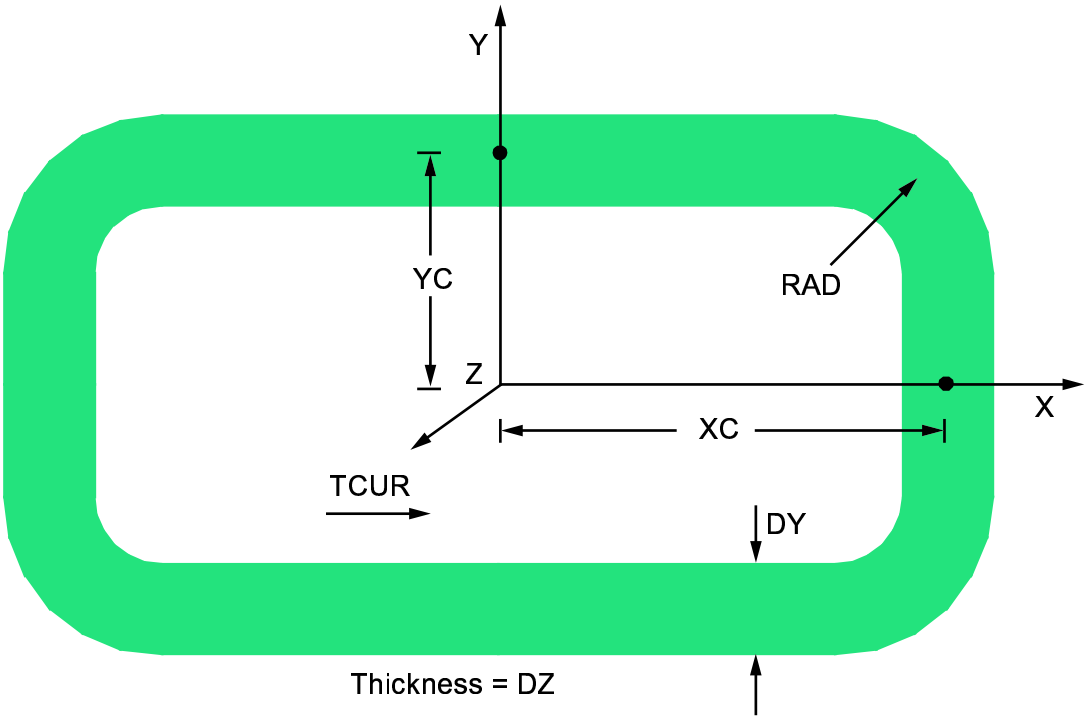
--, --
Unused fields

Cname
An alphanumeric name assigned as a component name to the group of SOURC36 elements created by the command macro. Cname must be enclosed in single quotes in the **RACE** command line. Cname may be up to 8 characters, beginning with a letter and containing only letters, numbers, and underscores. Component names beginning with an underscore (e.g., _LOOP) are reserved for use by ANSYS and should be avoided. If blank, no component name is assigned.

Notes

RACE invokes an ANSYS macro which defines a "racetrack" current source in the working plane coordinate system. The current source is generated from bar and arc source primitives using the SOURC36 element (which is assigned the next available element type number). The macro is valid for use in 3-D magnetic field analysis using a scalar potential formulation. Current flows in a counterclockwise direction with respect to the working plane.

The diagram below shows you a racetrack current source.



Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>Racetrack Coil
Main Menu>Preprocessor>Modeling>Create>Racetrack Coil
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>Racetrack Coil

---

RADOPT, *FLUXRELX*, *FLUXTOL*, *SOLVER*, *MAXITER*, *TOLER*, *OVERRLEX*
Specifies Gauss-Seidel Radiosity Solver options.

SOLUTION: Radiosity

MP ME <> <> <> PR <> <> <> PP ED

FLUXRELX
Relaxation factor applied to the heat flux. Defaults to 0.1.

FLUXTOL
Convergence tolerance for radiation flux. Defaults to 0.0001 for ANSYS Thermal radiation analysis and FLOTRAN surface radiation analysis.

SOLVER
Choice of solver for radiosity calculation:

0
Iterative solver (default).

1
Direct solver.

MAXITER
Maximum number of iterations for iterative solver (*SOLVER* = 0). Defaults to 1000.

TOLER
Convergence tolerance for the iterative solver (*SOLVER* = 0). Defaults to 0.1.

OVERRLEX
Over relaxation factor applied to the iterative solver (*SOLVER* = 0). Defaults to 0.1.

Menu Paths

Main Menu>Preprocessor>Radiation Opts>Solution Opt
Main Menu>Radiation Opt>Radiosity Meth>Solution Opt
Main Menu>Solution>Radiation Opts>Solution Opt

RALL

Calculates solver statistics and run time estimates.

RUNSTATS: Run Statistics Estimator
MP ME ST <> <> PR EM <> FL PP ED

Notes

Calculates solver statistics and estimates. The **RALL** command is a convenience command for obtaining all of the following: run time estimates [**RTIMST**], the wavefront statistics and memory requirements [**RWFRNT**], the file sizes estimates [**RFILSZ**], the memory statistics [**RMEMRY**], and the finite element model statistics [**RSTAT**].

Menu Paths

Main Menu>Run-Time Stats>All Statistics

RAPPND, *LSTEP*, *TIME*

Appends results data from the database to the results file.

POST1: Load Case Calculations
MP ME ST DY <> PR EM <> FL PP ED

LSTEP
Load step number to be assigned to the results data set. If it is the same as an existing load step number on the results file, the appended load step will be inaccessible. Defaults to 1.

TIME
Time value to be assigned to the results data set. Defaults to 0.0. A time value greater than the last load step should be used.

Notes

This command is typically used to append the results from a load case combination to the results file. See the **LCWRITE** command to create a separate load case file. Only summable and constant data are written to the results file by default; non-summable data are not written unless requested (**LCSUM** command). RAPPND should not be used to append results from a harmonic analysis.

Menu Paths

Main Menu>General Postproc>Write Results

RATE, *Option*

Specifies whether the effect of creep strain rate will be used in the solution of a load step.

SOLUTION: Analysis Options

MP ME ST <> <> <> <> <> <> PP ED

Option

Turns implicit creep analysis on or off.

0
(or OFF) Bypass creep analysis (default).

1
(or ON) Perform creep analysis.

Command Default

ANSYS automatically turns implicit creep OFF.

Notes

You must set *Option* = ON to perform an implicit creep analysis (**TB**,CREEP with *TBOPT* $\geq$ 1). For viscoplasticity/creep analysis, specifies whether or not to include the creep calculation in the solution of a load step. If *Option* = ON, ANSYS performs the creep calculation. Set an appropriate time for solving the load step using **TIME**,*TIME*.

Product Restrictions

This command works only when modeling *implicit* creep with either von Mises or Hill potentials.

When modeling implicit creep with von Mises potential, you can use the following elements: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209.

When modeling anisotropic creep (**TB**,CREEP with **TB**,HILL), you can also use the following elements: PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Strn Rate Effect
Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear
Main Menu>Solution>Load Step Opts>Nonlinear>Strn Rate Effect

/RATIO, *WN*, *RATOX*, *RATOY*

Distorts the object geometry.

GRAPHICS: Scaling

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

RATOX
Distort object in the window X direction by this factor (defaults to 1.0).

RATOY
Distort object in the window Y direction by this factor (defaults to 1.0).

Command Default

No distortion.

Notes

Distorts the object geometry in a particular direction. An example of this command's use would be to allow long narrow sections to be distorted to a more square area for better display visualization.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

RBE3, *Master*, *DOF*, *Slaves*, *Wtfact*

Distributes the force/moment applied at the master node to a set of slave nodes, taking into account the geometry of the slave nodes as well as weighting factors.

PREP7: Constraint Equations

MP ME ST DY <> PR EM <> <> PP ED

Master
Node at which the force/moment to be distributed will be applied. This node must be associated with an element for the master node to be included in the DOF solution.

DOF
Refers to the master node degrees of freedom to be used in constraint equations. Valid labels are: UX, UY, UZ, ROTX, ROTY, ROTZ, UXYZ, RXYZ, ALL

Slaves
The name of an array parameter that contains a list of slave nodes. Must specify the starting index number. ALL can be used for currently selected set of nodes.

Wtfact
The name of an array parameter that contains a list of weighting factors corresponding to each slave node above. Must have the starting index number. If not specified, the weighting factor for each slave node defaults to 1.

Notes

The force is distributed to the slave nodes proportional to the weighting factors. The moment is distributed as forces to the slaves; these forces are proportional to the distance from the center of gravity of the slave nodes times the weighting factors. Only the translational degrees of freedom of the slave nodes are used for constructing the constraint equations. Constraint equations are converted to distributed forces/moments on the slave nodes during solution.

Applying this command to a large number of slave nodes may result in constraint equations with a large number of coefficients. This may significantly increase the peak memory required during the process of element assembly. If real memory or virtual memory is not available, consider reducing the number of slave nodes.

As an alternative to the **RBE3** command, you can apply a similar type of constraint using contact elements and the internal multipoint constraint (MPC) algorithm. See Surface-based Constraints for more information.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Dist F/M at Mstr

RCON
Specifies "Real constants" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Real Constants

RDELE, *NSET1*, *NSET2*, *NINC*
Deletes real constant sets.

PREP7: Real Constants

MP ME ST DY <> PR EM <> FL PP ED

NSET1, *NSET2*, *NINC*

Delete real constant sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL, ignore *NSET2* and *NINC* and all real constant sets are deleted.

Notes

Deletes real constant sets defined with the **R** command.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Real Constants>Add/Edit/Delete
Main Menu>Preprocessor>Real Constants
Main Menu>Preprocessor>Real Constants>Add/Edit/Delete
Main Menu>Solution>Load Step Opts>Other>Real Constants>Add/Edit/Delete

---

REAL, *NSET*

Sets the element real constant set attribute pointer.

PREP7: Meshing
PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

NSET
Assign this real constant set number to subsequently defined elements (defaults to 1).

Command Default

NSET = 1.

Notes

Identifies the real constant set number to be assigned to subsequently defined elements. This number refers to the real constant set number (*NSET*) defined with the real constant sets [**R**]. Real constant set numbers may be displayed [**/PNUM**]. If the element type requires no real constants, this entry is ignored. Elements of different type should not refer to the same real constant set.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs
Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

---

REALVAR, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*

Forms a variable using only the real part of a complex variable.

POST26: Operations

MP ME ST DY <> PR EM <> <> PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA
Reference number of the variable to be operated on.

--, *--*
Unused fields.

Name
Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, *--*
Unused fields.

FACTA
Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

Notes

Forms a variable using only the real part of a variable. Used only with harmonic analyses (**ANTYPE**,HARMIC).

Complex variables are stored in two-column arrays with the real component stored in the first column and the imaginary component stored in the second column. This command extracts the value stored in the first column (i.e., real component). However with harmonic analyses, all variables are stored in two-column arrays as complex variables. If the variable is not complex, then the same value is stored in both columns. This command will extract the variable in the first column of the array, even if this variable is not the real component of a complex variable.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Real Part

RECTNG, *X1*, *X2*, *Y1*, *Y2*

Creates a rectangular area anywhere on the working plane.

PREP7: Primitives

MP ME ST DY <> PR EM <> FL PP ED

X1, *X2*
Working plane X coordinates of the rectangle.

Y1, *Y2*
Working plane Y coordinates of the rectangle.

Notes

The area will be defined with four keypoints and four lines. See the **BLC4** and **BLC5** commands for alternate ways to create rectangles.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Rectangle>By Dimensions

---

REDUCE, *NLOC*, *LENG*, *ELEM*

Defines a reducer in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

NLOC
Node where two straight pipes intersect at center of reducer. Defaults to previous run starting point.

LENG
Length of reducer (defaults to average pipe OD).

ELEM
Element number to be assigned to reducer (defaults to MAXEL + 1).

Notes

Defines a reducer (straight pipe element (PIPE16) with averaged specifications) in place of the intersection of two previously defined straight pipe elements in a piping run. See the PREP7 **RUN** command. Two new nodes are generated at the ends of the reducer. The two straight pipes are automatically "shortened" to meet the ends of the reducer. The reducer specifications and loadings are taken from the corresponding two straight pipes.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Define Pipes>Reducer

---

REFLCOEF, *Portin*, *Pvolt*, *Pang*, *Pdist*, *Vpathy*

Calculates the voltage reflection coefficient (REFLC), standing wave ratio (VSWR), and return loss (RL) in a COAX fed device; at postprocessing of an HF electromagnetic analysis.

POST1: Magnetics Calculations

MP ME <> <> <> <> EM <> <> PP ED

Portin
Port number of the excited port with a COAX mode excitation. (See the **HFPORT** command description for details.)

Pvolt
Port EMF (voltage) (magnitude) applied to the excited port.

Pang
Phase angle of the port EMF (voltage) (in degrees). Defaults to zero degrees.

Pdist
Propagation distance between the excited port and the evaluation point. Defaults to zero (evaluation at the excited port).

Vpathy
Path name defining a path between conducting walls of the coax waveguide at the specified propagation distance (*Pdist*) from the excited port. (See also the **PATH** command description.)

Notes

You must specify a path [**PATH** command] at the propagation distance location between conducting walls of the COAX waveguide for calculating the EMF (voltage). **REFLCOEF** returns the parameters REFLC, VSWR, RL, and REFANG (phase angle of the reflection coefficient).

To calculate the reflection coefficient, **REFLCOEF** uses total and incident EMF (voltage). It prints the resulting parameters to an output device and to your screen.

See magnetic macros for further details.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Port>Refl Coeff

/RENAME, *Fname1*, *Ext1*, --, *Fname2*, *Ext2*, --

Renames a file.

SESSION: Files

MP ME ST DY <> PR EM <> FL PP ED

Fname1
Name of file to be renamed and its directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

File name defaults to the current **Jobname**.

Ext1
Filename extension (8 character maximum).

--
Unused field

Fname2
Name of file to be renamed and its directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Fname2 defaults to *Fname1*.

Ext2
Filename extension (8 character maximum).

Ext2 defaults to *Ext1*.

--
Unused field

Notes

Renames a file. Ex: **/RENAME**,A,,,B renames file A to B in the same directory. **/RENAME**,A,DAT,,,INP renames file A.DAT to A.INP. On all systems, this command will overwrite any existing file named B. See the *ANSYS Operations*

Guide for details. Only ANSYS binary files should be renamed. Use **/SYS** and system renaming commands for other files.

Renaming across system partitions may be internally done by a copy and delete operation on some systems.

This command is valid only at the Begin Level.

Menu Paths

Utility Menu>File>File Operations>Rename

REORDER

Specifies "Model reordering" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Element Reorder>List Wave Lists
Utility Menu>List>Status>Preprocessor>Reorder Module

/REPLOT, *Label*

Automatically reissues the last display command for convenience.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Label

Controls the type of replot.

RESIZE

Issued internally when a graphics window resize occurs (Default).

FAST

Only applicable for 3-D devices that allow a fast redisplay for changes in the view characteristics only.

Notes

Reissues the last display command (**NPLOT**, **EPLOT**, **KPLOT**, **PLNSOL**, **PLVAR**, etc.), along with its parameters, for convenience. The current display specifications are used.

When the last display command is invalid in a particular processor, the use of the **/REPLOT** command is also invalid in that processor. However, if you attempt a **/REPLOT** and the last display command is invalid in the current processor, ANSYS produces an element display [**EPLOT**] instead, *as long as the last display command was* **PLNSOL**, **PLESOL**, *or* **PLDISP**. ANSYS performs this substitution of **/REPLOT** with **EPLOT** for your convenience.

For example, the **PLNSOL** command, which is used to display solution results as continuous contours, is a valid command in the general postprocessor [**/POST1**]. If you issue **PLNSOL** followed by **/REPLOT** while in the general postprocessor, the **/REPLOT** command effectively reissues your earlier **PLNSOL** command, along with its parameters. But if you then exit the general postprocessor, enter the preprocessor [**/PREP7**], and issue the **/REPLOT** command again, ANSYS internally issues **EPLOT** instead. This occurs because **PLNSOL** is not a valid command in the preprocessor.

When you click on one of the buttons on the **Pan, Zoom, Rotate** dialog box to manipulate the view of a model, the **/REPLOT** command is issued internally. Thus, the substitution of **/REPLOT** with **EPLOT** as described above may occur not only for the **PLNSOL**, **PLESOL**, and **PLDISP** results display commands, but also for operations that you perform with the **Pan, Zoom, Rotate** dialog box.

/REPLOT will not show boundary conditions if they are only applied to a solid model and the last display command (for example, **EPLOT**) displays the finite element model. To show boundary conditions, the following options are available:

- Issue **/REPLOT** after you issue the **SBCTRAN** command to transfer solid model boundary conditions to the finite element model.
- Issue **/REPLOT** after you issue a solid model display command (for example, **VPLOT**).

This command is valid in any processor (except as noted above).

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane
Main Menu>General Postproc>Path Operations>Delete Path>All Paths
Main Menu>General Postproc>Path Operations>Delete Path>By Name
Main Menu>General Postproc>Path Operations>Plot Paths
Main Menu>Preprocessor>Modeling>Create>Circuit>Scale Icon
Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane
Main Menu>Preprocessor>Path Operations>Delete Path>All Paths
Main Menu>Preprocessor>Path Operations>Delete Path>By Name
Main Menu>Preprocessor>Path Operations>Plot Paths
Utility Menu>Plot>Replot
Utility Menu>PlotCtrls>Style>Symmetry Expansion>Expansion by values

---

RESCONTROL, *Action*, *Ldstep*, *Frequency*, *MAXFILES*

Controls file writing for multiframe restarts.

SOLUTION: Misc Loads

MP ME ST <> <> PR <> <> <> PP ED

Action

Specify the command action.

DEFINE
Issuing the command will specify how frequently the **.R*nnn*** restart files are written for a load step (default).

FILE_SUMMARY
Issuing the command will print the substep and load step information for all **.R*nnn*** files for the current job name in the current directory. If this option is specified, all other arguments are ignored.

STATUS
Issuing the command will list the current status in the tables of restart control specified earlier by **RESCONTROL**.

Ldstep
Specify how the **.R*nnn*** files are written.

ALL
The **.R*nnn*** files are written at the same frequency for all load steps.

LAST
Write the **.R*nnn*** files for the last load step only (default).

N
Write the **.R*nnn*** files at the frequency indicated only for load step *N*. Other load steps will be written at the default frequency or at a frequency defined by a previous **RESCONTROL**,DEFINE,ALL,*Frequency* command.

NONE
No multiframe restart files (**.RDB** [restart database file], **.LDHI** [load history file], **.R*nnn***) will be created. If you specify this option, all other arguments will be ignored. This allows a restart to be done at the last or abort point using the same procedure as in ANSYS 5.5 or earlier (using the **.EMAT**, **.ESAV** or **.OSAV**, and **.DB** files).

Frequency
Frequency at which the **.R*nnn*** files are written.

NONE
Don't write any **.R*nnn*** files for this load step.

LAST
Write the **.R*nnn*** files for the last substep of the load step only (default).

N
If *N* is positive, write the **.R*nnn*** file every *N*th substep of a load step. If *N* is negative, write *N* equally spaced **.R*nnn*** files within a load step. Negative *N* is valid only when **AUTOTS**,ON.

MAXFILES
Maximum number of **.R*nnn*** files to save for *Ldstep*.

0
Do not overwrite any existing **.R*nnn*** files (default). The total maximum number of **.R*nnn*** files for one run is 999. If this number is reached before the analysis is complete, the analysis will continue but will no longer write any **.R*nnn*** files.

N
The maximum number of **.R*nnn*** files to keep for each load step. When *N* **.R*nnn*** files have been written for a load step, ANSYS will overwrite the first **.R*nnn*** file of that load step for subsequent substeps.

Command Default

If the **RESCONTROL** command is not issued during a structural analysis, the **.RDB** and **.LDHI** files will be written as described in Restarting an Analysis in the *ANSYS Basic Analysis Guide*. The **.R*nnn*** file will be written at the last substep of the last load step by default. A **.R*nnn*** file will also be written at the iteration prior to the abort point of the run if a **Jobname.ABT** file was used, or if the job terminates because of a failure to reach convergence or some other solution error. No information at the aborted substep will be saved to the **.R*nnn*** file.

Notes

This command sets up the restart parameters for a multiframe restart, which allows you to restart an analysis from any load step and substep for which there is a **.R*nnn*** file. You can do a multiframe restart only for nonlinear static and full transient structural analyses. (For linear static analyses, you can use a single frame restart.) For information on how to do a single frame or multiframe restart, and descriptions of the contents of the files used for each type of restart, see Restarting an Analysis in the *ANSYS Basic Analysis Guide*.

If you have many substeps for each load step, and are writing **.R*nnn*** files frequently, you may want to use *MAXFILES* to limit the number of **.R*nnn*** saved, since these files can fill up your disk quickly. You may specify *MAXFILES* and *Frequency* for individual load steps. These arguments will take on the default value or the value defined by **RESCONTROL**,,ALL,*Frequency*,*MAXFILES* if they are not explicitly defined for a specific load step.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Restart Control
Main Menu>Solution>Load Step Opts>Nonlinear>Restart Control

RESET

Resets all POST1 or POST26 specifications to initial defaults.

POST1: Set Up
POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Notes

Has the same effect as entering the processor the first time within the run. In POST1, resets all specifications to initial defaults, erases all element table items, path table data, fatigue table data, and load case pointers. In POST26, resets all specifications to initial defaults, erases all variables defined, and zeroes the data storage space.

Menu Paths

Main Menu>General Postproc>Reset
Main Menu>TimeHist Postpro>Reset Postproc

/RESET

Resets display specifications to their initial defaults.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Notes

Resets slash display specifications (**/WINDOW**, **/TYPE**, **/VIEW**, etc.) back to their initial default settings (for convenience). Also resets the focus location to the geometric center of the object.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Reset Plot Ctrls

RESP, *IR*, *LFTAB*, *LDTAB*, *ITYPE*, *RATIO*, *DTIME*, *TMIN*, *TMAX*

Generates a response spectrum.

POST26: Special Purpose

MP ME ST <> <> PR <> <> <> PP ED

IR

Arbitrary reference number assigned to the response spectrum results (2 to NV [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with these results.

LFTAB

Reference number of variable containing frequency table (created with **FILLDATA** or **DATA** command). The frequency table defines the number and frequency of oscillating systems used to determine the response spectrum. The frequency interval need not be constant over the entire range. Frequencies must be input in ascending order.

LDTAB

Reference number of variable containing the displacement time-history.

ITYPE

Defines the type of response spectrum to be calculated:

0 or 1
: Displacement.

2
: Velocity.

3
: Acceleration response spectrum.

RATIO

Ratio of viscous damping to critical damping (input as a decimal number).

DTIME

Integration time step (ITS) size used in the numerical integration scheme. This value should be equal to or greater than that actually used in the initial transient analysis. With **ANTYPE**,TRANS data, *DTIME* defaults to a value of 1/((20)(FMAX)), where FMAX is the highest frequency in *LFTAB*. For reduced linear transient dynamic (**ANTYPE**,TRANS) displacement pass data, the ITS read from the file (previously input for *DTIME* in the first load step of the reduced linear transient dynamic (**ANTYPE**,TRANS) analysis) is used for the default.

TMIN, *TMAX*

Specifies a subset of the displacement-time history to be used in the response spectrum calculation. Defaults to the full time range.

Notes

Generates a response spectrum from displacement time-history and frequency data.

The ANSYS modal analysis (**ANTYPE**,MODAL) may be followed by a spectrum analysis (**ANTYPE**,SPECTR). This analysis requires a response spectrum input of up to 20 points. This input may be determined from the response spectrum printout or display of this command and input to the modal analysis (by hand). The response spectrum generator uses the displacements from either a full or reduced transient dynamic (**ANTYPE**,TRANS) analysis. If a response spectrum is to be calculated from a given displacement time-history, the displacement time-history may be input to a single one-element reduced linear transient dynamic (**ANTYPE**,TRANS) analysis, so that the calculated output (which should be the same as the input) will be properly located on the file.

The response spectrum is defined as the maximum response of single degree of freedom systems of varying frequency (or period) to a given input support excitation. The equation describing the response of the system in terms of the relative displacement (X) is:

$$\ddot{\bar{X}} + 2\xi_n \omega_n \dot{\bar{X}} + \omega_n^2 \bar{X} = -X_o$$

where:

Ω_n = natural frequency of the system, $\sqrt{k/m}$
ξ_n = ratio of viscous damping to critical damping, c/c_{cr}
X_o = ground displacement

The solution of this equation for the maximum response, $\bar{X}_{max}$, at various frequencies results in the spectral response curve. See the *ANSYS, Inc. Theory Reference* for calculation details.

Calculations are based on a numerical integration scheme with the displacement time-history data from the file as the input ground-forcing function. The integration time step (argument *DTIME* on the **RESP** command) and the damping coefficient (argument *RATIO*) are constant over the frequency range. The number of calculations done per displacement spectral response curve is the product of the number of input solution points (*TMAX*-*TMIN*)*DTIME* and the number of oscillating systems (frequencies located in variable *LFTAB*). Input solution points requested (by *DTIME* and the frequency range) at a time not corresponding to an actual displacement solution time on the file are read from the next available time. The user has the option of calculating either a displacement, velocity, or acceleration spectral response.

Menu Paths

Main Menu>TimeHist Postpro>Generate Spectrm

RESUME, *Fname*, *Ext*, --, *NOPAR*, *KNOPLOT*

Resumes the database from the database file.

DATABASE: Set Up

MP ME ST DY <> PR EM EH FL PP ED

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext

Filename extension (8 character maximum).

The extension defaults to DB if *Fname* is blank.

--

Unused field

NOPAR

Parameter resume key:

0

All data in the database, *including* the scalar parameters, are replaced with the data saved on **File.DB** (default).

1

All data in the database, *except* the scalar parameters, are replaced with the data saved on **File.DB**.

Note — This option should *not* be used if array parameters are defined, since existing array parameters might be redefined with arbitrary values. See **PARSAV** and **PARRES** for a more general method of preventing the replacement of both scalar and array parameters.)

KNOPLOT

If equal to 1, will suppress automatic plot. Otherwise, if the GUI is on and this **RESUME** command was not read from a file, the selected elements from *Fname* are plotted. (If there are no selected elements, selected nodes are plotted. If no nodes, volumes; if no volumes, areas; if no areas, lines; if no lines, keypoints. If there are no selected keypoints, the screen is erased.)

Notes

Using **RESUME**, you can resume a database file into the same version of ANSYS that the file was created in. As long as you are resuming the file into the ANSYS version that the file was created in, you do not need to manipulate or modify the file in any way. Also, although not guaranteed, you can usually resume a database file created in the previous version of ANSYS into the current version. For example, you can probably resume an ANSYS 5.2 database file into ANSYS 5.3 without encountering problems. However, ANSYS is not expected to resume an ANSYS 5.2 database file into ANSYS 5.4 or later.

RESUME causes the database file (**File.DB**) to be read, thereby resetting the database (including any geometry settings) either a) as it was at the last **SAVE** command or b) as saved with the last **/EXIT** command, whichever was last. For multiple load step analyses, since only the data for one load step at a time may reside in the database, the load step data restored to the database will correspond to the load step data written when the save was done.

If the database file was saved [**SAVE**] in another ANSYS product, it may contain element type and **KEYOPT** specifications which are invalid in the "resuming" product. Immediately after the database resume is completed, you should redefine these invalid element types and **KEYOPT** settings to valid ones [**ET**, **KEYOPT**].

RESUME checks your database for mixed mode (both SMOOTH and FACETED) geometry. ANSYS will notify you if both types of geometry are found in the database. You can continue if you get this message. However, you may encounter problems during Boolean operations. If you do not want to continue the analysis with this database, you must either recreate or reimport the geometry.

This command is valid in any processor. If used in SOLUTION, this command is valid only within the first load step.

Menu Paths

Utility Menu>File>Resume from
Utility Menu>File>Resume Jobname.db

REXPORT, *Target*, --, --, *LSTEP*, *SBSTEP*, *Fname*, *Ext*, --

Exports displacements from an implicit run to ANSYS LS-DYNA.

SOLUTION: Explicit Dynamics

MP ME ST <> <> PR <> <> <> PP ED

Target
The type of analysis run to which displacements are exported.

OFF
Ignore initial displacements.

DYNA
Get initial displacements from an earlier implicit (ANSYS) run and export to an explicit ANSYS LS-DYNA run (Default).

--, --
Unused fields.

LSTEP
Load step number of data to be exported. Defaults to the last load step.

SBSTEP
Substep number of data to be exported. Defaults to the last substep.

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name does not have a default; you must specify a name. It CANNOT be the current **Jobname**.

Ext
Filename extension (8 character maximum).

The extension must be an RST extension (default). Currently, only structural results are allowed.

--
Unused field

Notes

This command exports the displacements, rotations, and temperatures calculated in an ANSYS implicit analysis into the `drelax' file, which is subsequently read in by ANSYS LS-DYNA when a dynamic relaxation or stress initialization is conducted [**EDDRELAX**].

This command is not written to the **Jobname.CDB** file when the **CDWRITE** command is issued.

Menu Paths

Main Menu>Preprocessor>LS-DYNA Options>Constraints>Read Disp
Main Menu>Solution>Constraints>Read Disp

RFILSZ

Estimates file sizes.

RUNSTATS: Run Statistics Estimator

MP ME ST <> <> PR EM <> <> PP ED

Notes

Gives file size estimates for **File.ESAV**, **File.EMAT**, **File.EROT**, **File.TRI**, **File.FULL**, **File.RST**, **File.RTH**, and **File.RMG**. File sizes are given in megabytes (MB). These file estimates are for the solution of the model that currently resides in the database. This command will cause reordering [**WAVES**] if reordering was not already done on the model. See File Management and Files in the *ANSYS Basic Analysis Guide* for file descriptions.

Menu Paths

Main Menu>Run-Time Stats>Individual Stats

RFORCE, *NVAR*, *NODE*, *Item*, *Comp*, *Name*

Specifies the total reaction force data to be stored.

POST26: Set Up

MP ME ST DY <> PR EM <> <> PP ED

NVAR
Arbitrary reference number assigned to this variable (2 to NV [**NUMVAR**]). Overwrites any existing results for this variable.

NODE
Node for which data are to be stored. If *NODE* = P, graphical picking is enabled (valid only in the GUI).

Item
Label identifying the item. Valid item labels are shown in the table below. Some items also require a component label.

Comp
Component of the item (if required). Valid component labels are shown in the table below.

Name

Thirty-two character name identifying the item on printouts and displays. Defaults to an eight character label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Notes

Defines the total reaction force data (static, damping, and inertial components) to be stored from single pass (**ANTYPE**,STATIC or TRANS) solutions or from an expansion pass of reduced two-pass (**ANTYPE**,HARMIC or TRANS) solutions.

RFORCE - Valid Item and Component Labels

Valid item and component labels for node results are:

Item	Comp	Description
F	X,Y,Z	X, Y, or Z structural force.
M	X,Y,Z	X, Y, or Z structural moment.
HEAT[1]		Heat flow.
FLOW		Fluid flow.
AMPS		Current flow.
FLUX		Magnetic flux.
VF	X,Y,Z	X, Y, or Z fluid force component.
CSG	X,Y,Z	X, Y, or Z magnetic current segment component.
VLTG		Voltage drop
CURT		Current
CHRG		Charge

1. For **SHELL131** and **SHELL132** elements with KEYOPT(3) = 0 or 1, use the labels HBOT, HE2, HE3, . . ., HTOP instead of HEAT.

Menu Paths

Main Menu>TimeHist Postpro>Define Variables
Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

/RGB, *Kywrd*, *PRED*, *PGRN*, *PBLU*, *N1*, *N2*, *NINC*, *NCNTR*

Specifies the RGB color values for indices and contours.

POST26: Set Up

MP ME ST DY <> PR EM <> <> PP ED

Kywrd

Determines how RGB modifications will be applied.

INDEX

Specifies that subsequent color values apply to ANSYS color indices (0-15).

CNTR
Specifies that subsequent color values apply to contours (1-128). Applies to C-option devices only (i.e. X11C or Win32C).

PRED
Intensity of the color red, expressed as a percentage.

PGRN
Intensity of the color green, expressed as a percentage.

PBLU
Intensity of the color blue, expressed as a percentage.

N1
First index (0-15), or contour (1-128) to which the designated RGB values apply.

N2
Final index (0-15), or contour (1-128) to which the designated RGB values apply.

NINC
The step increment between the values *N1* and *N2* determining which contours or indices will be controlled by the specified RGB values.

NCNTR
The new maximum number of contours (1-128).

Notes

Issuing the **/CMAP** command (with no filename) will restore the default color settings.

Menu Paths

Utility Menu>PlotCtrls>Redirect Plots>To GRPH File
Utility Menu>PlotCtrls>Redirect Plots>To HPGL File
Utility Menu>PlotCtrls>Redirect Plots>To HPGL2 File
Utility Menu>PlotCtrls>Redirect Plots>To PSCR File

RIGID, *Dof1*, *Dof2*, *Dof3*, *Dof4*, *Dof5*, *Dof6*

Specifies known rigid body modes (if any) of the model.

SOLUTION: Dynamic Options

MP ME ST <> <> PR <> <> <> PP ED

Dof1, *Dof2*, *Dof3*, *Dof4*, *Dof5*, *Dof6*
Up to six global Cartesian directions of the rigid modes. For a completely free 2-D model, use ALL or UX, UY, ROTZ. For a completely free 3-D model, use ALL or UX, UY, UZ, ROTX, ROTY, ROTZ. For a constrained model, use UX, UY, UZ, ROTX, ROTY, or ROTZ, as appropriate, to specify each and every unconstrained direction which exists in the model (not specifying every direction may cause difficulties in extracting the modes). Use NONE to force the subspace iteration calculation of all rigid body modes. If the structure has no constraints, the label ALL is assumed (unless substructures are present).

Command Default

Any rigid body modes are calculated via subspace iteration.

Notes

Specifies known rigid body modes (if any) of the model. Applies only to modal analyses with subspace iteration [**MODOPT**,SUBSP]. Rigid body modes specified to the program with this command are not calculated via subspace iteration, resulting in a faster solution. Any rigid body modes specified must be permitted by the applied displacement constraints (i.e., do not specify a rigid body mode in a constrained direction). Reissue the command to redefine the specification. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

RIMPORT, *Source*, *Type*, *Loc*, *LSTEP*, *SBSTEP*, *Fname*, *Ext*, --, *SPSCALE*, *MSCALE*

Imports initial stresses from an explicit dynamics run into ANSYS.

SOLUTION: FE Body Loads

MP ME ST <> <> PR <> <> <> PP ED

Source
The type of analysis run from which stresses are imported.

OFF
Ignore initial stresses.

DYNA
Get initial stresses from an earlier explicit (ANSYS LS-DYNA) run (default).

Type
Type of data imported. Note that this is an ANSYS-defined field; the only valid value is STRESS.

Loc
Location where the data is imported. Note that this is an ANSYS-defined field; the only valid value is ELEM (data imported at the element integration points).

LSTEP
Load step number of data to be imported. Defaults to the last load step.

SBSTEP
Substep number of data to be imported. Defaults to the last substep.

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name does not have a default; you must specify a name. It CANNOT be the current **Jobname**.

Ext
Filename extension (8 character maximum).

The extension must be an RST extension (default).

- -
Unused field

SPSCALE
Stabilization factor. This factor is used in a springback analysis to scale (up or down) the initial stiffness of the applied spring. No default; input a value only if you want to activate stabilization. If *SPSCALE* is blank, stabilization is not activated.

MSCALE
Acceptable stabilization stiffness (defaults to 1.0×10^{-4}). In a springback analysis, iterations will stop when the applied spring stiffness comes down to this value. *MSCALE* is not used if *SPSCALE* is blank.

Notes

This command imports initial stress information into ANSYS from an earlier explicit (ANSYS LS-DYNA) run. The stress state from SHELL163 and SOLID164 elements in the explicit analysis is imported to the corresponding SHELL181 and SOLID185 implicit elements. For the shell elements, the current shell element thickness is also imported. This command is valid only before the first **SOLVE** command of the implicit analysis (which comes after the explicit analysis) and is ignored if issued after subsequent **SOLVE** commands (that is, stresses will not be re-imported).

RIMPORT is typically used to perform springback analysis of sheet metal forming. We recommend that you use SHELL163 elements in the explicit analysis with 3 to 5 integration points through the thickness. This ensures that the through-thickness stress distribution is transferred accurately to the SHELL181 elements. If more than 5 integration points are used, ANSYS imports resultants (forces and moments) to the SHELL181 elements. This implies that linearization of the through-thickness stress distribution is assumed in SHELL181 elements. If SHELL163 uses full integration in the shell plane, stress and thickness data are averaged and then transferred. For the solid elements, the stress at the SOLID164 element centroid is transferred to the SOLID185 element centroid. If SOLID164 has full integration, the stress is averaged and then transferred.

When the *SPSCALE* argument is specified, artificial springs with exponentially decaying stiffness (as a function of iterations) are applied. This technique is recommended only for those cases in which there are severe convergence difficulties. In general, you should first attempt a springback analysis without using the stabilization factors *SPSCALE* and *MSCALE*. (For more information on springback stabilization, see the *ANSYS LS-DYNA User's Guide*.)

This command is not written to the **Jobname.CDB** file when the **CDWRITE** command is issued. Further, the **RIMPORT** information is not saved to the database; therefore, the **RIMPORT** command must be reissued if the database is resumed.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Other>Import Stress
Main Menu>Solution>Define Loads>Apply>Structural>Other>Import Stress

RITER, *NITER*

Supplies an estimate of the number of iterations for time estimates.

RUNSTATS: Run Statistics Estimator

MP ME ST <> <> PR EM <> <> PP ED

NITER

Estimated number of iterations (or load steps in a linear, static analysis) for the analysis (defaults to 1).

Command Default

One iteration.

Notes

This estimate will be used by the program to calculate estimated run times [**RTIMST**].

Menu Paths

Main Menu>Run-Time Stats>Iter Setting

RLIST, *NSET1*, *NSET2*, *NINC*

Lists the real constant sets.

PREP7: Real Constants

MP ME ST DY <> PR EM <> FL PP ED

NSET1, *NSET2*, *NINC*

List real constant sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL (default), ignore *NSET2* and *NINC* and list all real constant sets [**R**].

Notes

The real constant sets listed contain only those values specifically set by the user. Default values for real constants set automatically within the various elements are not listed.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Properties>All Real Constants
Utility Menu>List>Properties>Specified Real Constants

RMALIST

Lists all defined master nodes for a ROM method.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

Menu Paths

Main Menu>ROM Tool>Setup>Master Nodes>List

RMANL, *Fname*, *Ext*, --, *Dimn*, *Oper*

Assigns model database, dimensionality, and operating direction for the ROM method.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

Fname
Database file name and directory path (248 characters maximum, including directory). The file name defaults to **Jobname**.

Ext
File extension (8 character maximum). The extension defaults to db.

--
Unused field

Dimn
Model dimensionality:

2
2-D models

3
3-D Models

Oper
Primary operating direction:

X
direction

Y
direction

Z
direction

Notes

Required Inputs:

Model database containing a structural physics file, title “STRU”.
Model database containing an electrostatic physics file, title “ELEC”.

Model database containing an area or volume component of the electrostatic domain to be morphed, title "AIR".
Model database containing a node component of the neutral plane nodes, named "NEUN".
Model database containing node components of conductors, named "CONDi", where "i" is the conductor number.

Menu Paths

Main Menu>ROM Tool>Setup>Model Features

RMASTER, *Node*, *Lab*

Defines master nodes for the ROM method.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

Node
Node number at which master degree of freedom is defined If *Node* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Lab
Valid labels are "ADD" (default) and "DEL".

Notes

Defines master nodes for the ROM. Master nodes are used to track the total displacement of a structure in the operating direction [**RMANL**]. They may be used as attachment points for 1-D structural elements during a ROM use pass via the UX degree of freedom.

Menu Paths

Main Menu>ROM Tool>Setup>Master Nodes>Define
Main Menu>ROM Tool>Setup>Master Nodes>Delete

RMCAP, *RefName*, *C1*, *C2*

Defines lumped capacitance pairs between conductors C1 and C2 for a ROM method.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

RefName
Reference name for capacitance pair definition.

C1
First conductor (between 1 and 5).

C2
Second conductor (between 1 and 5).

Notes

For a capacitance definition between conductor *C1* and *C2*, node components COND%C1% and COND%C2% (see **CM** command) must be present containing the conductor nodes. If *C1* and *C2* are blank, the capacitance definition with *RefName* will be deleted. (For example, if *C1* = 1, and *C2* = 2, then node components COND1 and COND2 must be defined).

Menu Paths

Main Menu>ROM Tool>Setup>Capacitances>Define>All Capacitances
Main Menu>ROM Tool>Setup>Capacitances>Define>Single Capacitance
Main Menu>ROM Tool>Setup>Capacitances>Delete

RMCLIST

Lists all lumped capacitance pairs defined.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

Menu Paths

Main Menu>ROM Tool>Setup>Capacitances>Define>List
Main Menu>ROM Tool>Setup>Capacitances>List

RMEMRY

Prints memory statistics for the current model.

RUNSTATS: Run Statistics Estimator

MP ME ST <> <> PR EM <> <> PP ED

Notes

Memory statistics include work space usage, database size, binary input/output buffers, and available ANSYS scratch space. Also the maximum available static wavefront that will fit in the available ANSYS scratch space is displayed. The memory statistics are displayed in units of kilobytes (KB) or megabytes (MB).

Menu Paths

Main Menu>Run-Time Stats>Individual Stats

RMFLVEC

Writes eigenvectors of fluid nodes to a file for use in damping parameter extraction.

POST1: Special Purpose

MP ME <> <> <> <> <> <> <> PP ED

Notes

RMFLVEC extracts the modal information from the modal results file for all nodes specified in a node component called 'FLUN'. This component should include all nodes which are located at the fluid-structural interface. Mode shapes, element normal orientation, and a scaling factor are computed and stored in a file **Jobname.EFL**. For damping parameter extraction, use the **DMPEXT** command macro. See Chapter 16, " Thin Film Analysis" for more information on thin film analyses.

FLUID136 and FLUID138 are used to model the fluid interface. Both the structural and fluid element types must be active. The fluid interface nodes must be grouped into a component 'FLUN'. A results file of the last modal analysis must be available.

Menu Paths

Main Menu>General Postproc>ThinFilm>Extract Eigv
Main Menu>Solution>ThinFilm>DampExtract>Eigenfrequency
Main Menu>Solution>ThinFilm>DampExtract>Frequency Range
Main Menu>Solution>ThinFilm>RayleighDamp

RMLVSCALE, *Nload*, *Fact1*, *Fact2*, *Fact3*, *Fact4*, *Fact5*

Defines element load vector scaling for a ROM use pass.

REDUCED ORDER MODELING: Use Pass

MP <> <> <> <> <> <> <> <> PP ED

Nload
Total number of load cases to be considered within a ROM use pass. If *Nload* = "DELETE", all defined load vectors are deleted.

Fact1, *Fact2*, *Fact3*, *Fact4*, *Fact5*
Scale factors applied to load vectors (maximum 5). Defaults to 0.

Notes

Specifies the element load scale factor applied to a ROM analysis use pass. Element load vectors are extracted from a Static Analysis using the **RMNDISP** command. Up to 5 element load vectors may be scaled and applied to a ROM use pass.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Load Vector>For ROM
Main Menu>Preprocessor>Loads>Define Loads>Delete>Load Vector>For ROM
Main Menu>Solution>Define Loads>Apply>Load Vector>For ROM
Main Menu>Solution>Define Loads>Delete>Load Vector>For ROM

RMMLIST

Lists all mode specifications for the ROM method.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

Menu Paths

Main Menu>ROM Tool>Mode Selection>List

RMMRANGE, *Mode*, *Key*, *Min*, *Max*, *Nstep*, *Damp*, *Scale*

Defines and edits various modal parameters for the ROM method.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

Mode
Mode number. Must be lower or equal to the number of modes extracted via the **RMNEVEC** command.

Key
Mode classification key. Valid keys are:

DOMINANT
Dominant mode

RELEVANT
Relevant mode

UNUSED
Unused mode. Do not consider mode in ROM.

Min
Lower bound for fit range of mode.

Max
Upper bound for fit range of mode.

Nstep
Number of equidistant steps in fit range of mode.

Damp
Modal damping factor. Defaults to 0.05.

Scale
Modal scaling factor.

Notes

When selected manually (**RMMSELECT**), modes must be classified as dominant, relevant, or unused. Dominant modes (*Key* = "DOMINANT") are basis functions with large amplitudes. Relevant modes (*Key* = "RELEVANT") are influenced by the dominant modes but do not cause interactions among themselves due to the small amplitude. This assumption leads to essential speed up of the sample point generator (see **RMSMPLE**).

Menu Paths

Main Menu>ROM Tool>Mode Selection>Edit

RMMSELECT, *Nmode*, *Method*, *Dmin*, *Dmax*

Selects modes for the ROM method.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

Nmode
Total number of modes to be selected

Method
Method for mode selection. Valid methods are:

TMOD
Automated selection using a test load. TMOD must be enclosed in single quotes.

NMOD
First *Nmode* eigenmodes. NMOD must be enclosed in single quotes.

Dmin
Lower bound for total deflection range.

Dmax
Upper bound for total deflection range.

Notes

Select pertinent modes for use in a ROM. Pertinent mode selection may be enhanced by using the deflection state of the structure representative of the operating nature of the device (*Method* = TMOD). A static analysis with an applied Test Load may be used. The test load displacements must be extracted at the neutral plane of the device (if the device is stress-stiffened), or at any plane of the device (non-stress-stiffened). A node component "NEUN" must be defined for the plane of nodes, and the displacements extracted using the **RMNDISP** command prior to issuing this command. If *Method* = NMOD, use the first *Nmode* eigenmodes to select the pertinent modes for the ROM tool. Only those modes are selected that act in the operating direction of the structure [**RMANL**].

Required Input Files

jobname.evx, **jobname.evy**, **jobname.evz**, **jobname.evn**, **jobname.evl**

Optional Input File

Test load and element load neutral plane displacement files: **jobname.tld**, **jobname.eld**

Menu Paths

Main Menu>ROM Tool>Mode Selection>Select

RMNDISP, *LoadT*, *Loc*

Extracts neutral plane displacements from a test load or element load solution for the ROM method.

REDUCED ORDER MODELING: Preparation

MP <> <> <> <> <> <> <> <> PP ED

LoadT

Load type. Load type must be an alphanumeric string enclosed in single quotes. Valid load types are 'TLOAD' for the test load and 'ELOAD' for the element load.

Loc

Determines whether file will be overwritten or appended. Valid labels are 'WRITE' or 'APPEND'. Defaults to 'WRITE' for test load.

Notes

This command extracts the displacements at a neutral plane of a model. If *LoadT* = 'TLOAD', extract displacements for a test load on a structure that represents the expected deflection state. A test load is used to assist in the automatic mode selection for the ROM mode characterization. If *LoadT* = 'ELOAD', extract the neutral plane displacements for an element load that will be used in the use pass of a ROM analysis. Typical element loads are gravity, and pressure loading. The element loads may be scaled [**RMLVSCALE**] during the use pass.

The command requires a node component named “NEUN” to be defined. These nodes represent the nodes at the neutral plane of a structure (in the case of a stress-stiffened structure), or at any plane in the structure (non stress-stiffened case).

For *LoadT* = 'TLOAD', node displacements are written to the file **jobname.tld**. For *LoadT* = 'ELOAD', node displacements are written to the file **jobname.eld**. Up to 5 element load cases may be written to the file **jobname.eld**.

This command is only valid in POST1.

Output Files

For a test load **jobname.tld**
For an element load **jobname.eld**

Menu Paths

Main Menu>General Postproc>ROM Operations>Extract NP DISP

RMNEVEC

Extracts neutral plane eigenvectors from a modal analysis for the ROM method.

REDUCED ORDER MODELING: Preparation

MP <> <> <> <> <> <> <> <> PP ED

Notes

This command extracts the eigenvectors at a neutral plane of a model from a modal analysis. The modal analysis must have expanded modes [**MXPAND**] in order to process the data. Only the first 9 modes are considered. The command requires a node component named “NEUN” to be defined. These nodes represent the nodes at the

neutral plane of a structure (in the case of a stress-stiffened structure), or at any plane in the structure (non stress-stiffened case).

This command is only valid in POST1.

Output Files

jobname.evx, **jobname.evy**, **jobname.evz**, **jobname.evn**, **jobname.evl**

Menu Paths

Main Menu>General Postproc>ROM Operations>Extract NP Eigv

RMODIF, *NSET*, *STLOC*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

Modifies real constant sets.

PREP7: Real Constants

MP ME ST DY <> PR EM <> FL PP ED

NSET
Existing set to be modified.

STLOC
Starting location in table for modifying data. For example, if *STLOC* = 1, data input in the *V1* field is the first constant in the set. If *STLOC* = 7, data input in the *V1* field is the seventh constant in the set, etc. Must be greater than zero.

V1
New value assigned to constant in location *STLOC*. If zero (or blank), a zero value will be assigned.

V2, *V3*, *V4*, *V5*, *V6*
New values assigned to constants in the next five locations. If blank, the value remains unchanged.

Notes

Allows modifying (or adding) real constants to an existing set [**R**] at any location, excluding table inputs.

This command is also valid in SOLUTION.

Menu Paths

This command cannot be accessed from a menu.

RMORE, *R7*, *R8*, *R9*, *R10*, *R11*, *R12*

Adds real constants to a set.

PREP7: Real Constants

MP ME ST DY <> PR EM <> FL PP ED

R7, *R8*, *R9*, *R10*, *R11*, *R12*
Add real constants 7 to 12 (numerical values or table names) to the most recently defined set.

Notes

Adds six more real constants to the most recently defined set. Repeat the **RMORE** command for constants 13 to 18, again for 19-24, etc.

If using table inputs (SURF151, SURF152, FLUID116, CONTA171, CONTA172, CONTA173, CONTA174, and CONTA175 only), enclose the table name in % signs (e.g., *%tabname%*).

When copying real constants to new sets, ANSYS recommends that you use the command input. If you do use the GUI, restrict the real constant copy to only the first six real constants (real constants seven and greater will be incorrect for both the master and copy set).

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Rotary
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Trans
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Transducer>ElecMech
Main Menu>Preprocessor>Modeling>Create>Circuit>Edit Real Cnst

---

RMPORDER, *Ord1*, *Ord2*, *Ord3*, *Ord4*, *Ord5*, *Ord6*, *Ord7*, *Ord8*, *Ord9*

Defines polynomial orders for ROM functions.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

Ord1, *Ord2*, *Ord3*, *Ord4*, *Ord5*, *Ord6*, *Ord7*, *Ord8*, *Ord9*

Polynomial orders for modes. *Ordi* specifies the polynomial order for mode*i*. Modes are ordered as extracted from a modal analysis using the **RMNEVEC** command. Defaults to 0 if mode *i* is unused; default to *nstep(i) - 1* for dominant or relevant modes, where *nstep(i)* is the number of equidistant steps in fit range of mode *i*. *nstep(i)* is automatically set by **RMMSELECT** or modified by the **RMMRANGE** command.

Menu Paths

Main Menu>ROM Tool>Resp Surface>Poly Order

RMRESUME, *Fname*, *Ext*, --

Resumes ROM data from a file.

REDUCED ORDER MODELING: Set Up

MP <> <> <> <> <> <> <> <> PP ED

Fname

Name and directory path of the ROM database file (248 character maximum). Default to **Jobname**.

Ext

Extension of the ROM database file. Default to .rom.

- -
Unused field

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Preprocessor>Loads>Analysis Type>ExpansionPass
Main Menu>Preprocessor>Loads>Analysis Type>New Analysis
Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>ROM>Database
Main Menu>ROM Tool>Rom Database>Resume
Main Menu>Solution>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>ExpansionPass
Main Menu>Solution>Analysis Type>New Analysis

RMRGENERATE

Performs fitting procedure for all ROM functions to generate response surfaces.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

Notes

The fitting procedure uses modal analysis data and function data generated using the **RMSMPLE** command and specifications set forth in the **RMROPTIONS** command. The files **jobname_ijk.pcs** (modes i, j, k) will be generated containing the coefficients of the response surfaces. These files are needed for the ROM Use Pass along with a ROM data base file [**RMSAVE**].

Input Files

Strain energy and capacitance data file **jobname_ijk.dec**

Output Files

Response surface coefficients **jobname_ijk.pcs** (modes i, j, k)

Menu Paths

Main Menu>ROM Tool>Resp Surface>Fit Functions

RMROPTIONS, *RefName*, *Type*, *Invert*

Defines options for ROM response surface fitting.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

RefName
Reference name of ROM function to be fitted. Valid reference names are "SENE" for the strain energy of the structural domain and any capacitance reference name previously defined by means of **RMCAP** command for the electrostatic domain.

Type
Type of fitting function to be applied for regression analysis. Valid types are:

LAGRANGE
Lagrange type (default)

REDLAGRANGE
Reduced Lagrange type

PASCAL
Pascal type

REDPASCAL
Reduced Pascal type

Invert
Flag to specify whether data should be inverted prior to fitting.

0
Do not invert data (default for SENE)

1
Invert data input on*Refname* field (recommended for capacitance functions).

Notes

The objective of response surface fit is to compute an analytical expression for the strain energy and the capacitance as functions of modal amplitudes.

Menu Paths

Main Menu>ROM Tool>Resp Surface>Options

RMRPLOT, *RefName*, *Type*, *Mode1*, *Mode2*

Plots response surface of ROM function or its derivatives with respect to the dominant mode(s).

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

RefName
Reference name of ROM function. Valid reference names are "SENE" for the strain energy of the mechanical domain and any capacitance definition, previously defined by means of the **RMCAP** command, for the electrostatic domain.

Type
Type of data to be plotted. Valid types are:

FUNC
Response surface (default)

FIRST
First derivative of response surface with respect to *Mode1*.

SECOND
Second derivative of response surface with respect to *Mode1* and *Mode2*.

Mode1
First mode number (used for *Type* = "FIRST" and *Type* = "SECOND" only).

Mode2
Second mode number (used for *Type* = "SECOND" only).

Notes

The objective of response surface fit is to compute an analytical expression for the strain energy and the capacitance as functions of modal amplitudes. This command assumes that the coefficient files **jobnam_ijk.pcs** are available [**RMRGENERATE**]. Visualization of the response surface will help to evaluate the validity of the function fit.

Menu Paths

Main Menu>ROM Tool>Resp Surface>Plot

RMRSTATUS, *RefName*

Prints status of response surface for ROM function.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

RefName
Reference name of ROM function. Valid reference names are "SENE" for the strain energy of the mechanical domain and any capacitance reference names [**RMCAP**], for the electrostatic domain.

Menu Paths

Main Menu>ROM Tool>Resp Surface>Status

RMSAVE, *Fname*, *Ext*, --

Saves ROM data to file.

REDUCED ORDER MODELING: Set Up

MP <> <> <> <> <> <> <> <> PP ED

Fname
Name and directory path of the ROM database file. Default to **Jobname**.

Ext
Extension of the ROM database file. Default to .rom.

--
Unused field

Menu Paths

Main Menu>ROM Tool>Rom Database>Save

RMSMPLE, *Nlgeom*, *Cap*, *Seqslv*, *Eeqslv*

Runs finite element solutions and obtains sample points for the ROM method.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

Nlgeom

Specify whether a large or small deflection analysis is to be performed for the mechanical domain:

OFF (or 0)
Perform small deflection analysis (default).

ON (or 1)
Perform large deflection analysis.

Cap

Capacitance calculation method.

CHARGE
Compute capacitance based on the charge voltage relationship (default).

CMATRIX
Employ CMATRIX macro to calculate capacitance.

Seqslv

Solver for structural analysis:

SPARSE
Sparse direct equation solver (default).

PCG
Pre-conditioned Conjugate Gradient iterative equation solver.

Eeqslv

Solver for electrostatic analysis:

SPARSE
Sparse direct equation solver (default).

JCG
Jacobi Conjugate Gradient iterative equation solver.

ICCG
Incomplete Cholesky Conjugate Gradient iterative equation solver.

Notes

This command prepares and runs multiple finite element solutions on the Structural domain and the Electrostatic domain of a model to collect sample points of data for ROM response curve fitting. The command requires a model database [**RMANL**] and two Physics Files (Structural domain, titled "STRU" and an Electrostatic domain, titled "ELEC"; see **PHYSICS** command). Also required is a complete ROM database generated from the ROM Tools. The *Cap* = CHARGE method is preferred when capacitance to "infinity" is not required. Capacitance conductor pairs are defined by the **RMCAP** command.

Required Input

Model Database **filename.db**
ROM Database **jobname.rom**, **jobname.evx**, **jobname.evy**, **jobname.evz**

Output Files

Strain energy and capacitance data files **jobname_ijk.dec** (mode i, j, k).

Menu Paths

Main Menu>ROM Tool>Sample Pt Gen>Compute Points

RMUSE, *Option*, *Usefil*

Activates ROM use pass for ROM elements.

REDUCED ORDER MODELING: Use Pass

MP <> <> <> <> <> <> <> <> PP ED

Option
Type of data to be plotted. Valid types are:

1 or"ON"
Activates ROM use pass.

0 or"OFF"
Deactivates ROM use pass.

Usefil
Name of the reduced displacement file (**.rdsp**) created by the ROM Use Pass (required field only for the Expansion Pass).

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Preprocessor>Loads>Analysis Type>ExpansionPass
Main Menu>Preprocessor>Loads>Analysis Type>New Analysis
Main Menu>Solution>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>ExpansionPass
Main Menu>Solution>Analysis Type>New Analysis

RMXPORT

Exports ROM model to external VHDL-AMS simulator.

REDUCED ORDER MODELING: Generation Pass

MP <> <> <> <> <> <> <> <> PP ED

Notes

Use this command to generate all files necessary to run the ROM analysis in an external VHDL-AMS Simulator.

Output Files

VHDL files: **Initial.vhd**, **S_ams_ijk.vhd**, **Cxxx_ams_ijk.vhd**, **transducer.vhd**.

Menu Paths

Main Menu>ROM Tool>Export>VHDL-AMS

ROCK, *CGX*, *CGY*, *CGZ*, *OMX*, *OMY*, *OMZ*

Specifies a rocking response spectrum.

SOLUTION: Spectrum Options

MP ME ST <> <> PR <> <> <> PP ED

CGX, *CGY*, *CGZ*
Global Cartesian X, Y, and Z location of center of rotation about which rocking occurs.

OMX, *OMY*, *OMZ*
Global Cartesian angular velocity components associated with the rocking.

Notes

Specifies a rocking response spectrum effect in the spectrum (**ANTYPE**,SPECTR) analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Settings
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Settings

RPOLY, *NSIDES*, *LSIDE*, *MAJRAD*, *MINRAD*

Creates a regular polygonal area centered about the working plane origin.

PREP7: Primitives

MP ME ST DY <> PR EM <> FL PP ED

NSIDES
Number of sides in the regular polygon. Must be greater than 2.

LSIDE
Length of each side of the regular polygon.

MAJRAD
Radius of the major (or circumscribed) circle of the polygon. Not used if *LSIDE* is input.

MINRAD
Radius of the minor (or inscribed) circle of the polygon. Not used if *LSIDE* or *MAJRAD* is input.

Notes

Defines a regular polygonal area on the working plane. The polygon will be centered about the working plane origin, with the first keypoint defined at $\theta = 0°$. The area will be defined with *NSIDES* keypoints and *NSIDES* lines. See the **RPR4** and **POLY** commands for other ways to create polygons.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Circumscr Rad
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Inscribed Rad
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Side Length

---

RPR4, *NSIDES*, *XCENTER*, *YCENTER*, *RADIUS*, *THETA*, *DEPTH*

Creates a regular polygonal area or prism volume anywhere on the working plane.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

NSIDES
The number of sides in the polygon or prism face. Must be greater than 2.

XCENTER, *YCENTER*
Working plane X and Y coordinates of the center of the polygon or prism face.

RADIUS
Distance (major radius) from the center to a vertex of the polygon or prism face (where the first keypoint is defined).

THETA
Angle (in degrees) from the working plane X-axis to the vertex of the polygon or prism face where the first keypoint is defined. Used to orient the polygon or prism face. Defaults to zero.

DEPTH
The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the prism. If *DEPTH* = 0 (default), a polygonal area is created on the working plane.

Notes

Defines a regular polygonal area anywhere on the working plane or prism volume with one face anywhere on the working plane. The top and bottom faces of the prism are polygonal areas. See the **RPOLY**, **POLY**, **RPRISM**, and **PRISM** commands for other ways to create polygons and prisms.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Hexagon
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Octagon
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Pentagon
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Septagon
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Square
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Triangle
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Hexagonal
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Octagonal
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Pentagonal
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Septagonal
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Square
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Triangular

---

RPRISM, *Z1*, *Z2*, *NSIDES*, *LSIDE*, *MAJRAD*, *MINRAD*

Creates a regular prism volume centered about the working plane origin.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

Z1, *Z2*
Working plane Z coordinates of the prism.

NSIDES
Number of sides in the polygon defining the top and bottom faces of the prism. Must be greater than 2.

LSIDE
Length of each side of the polygon defining the top and bottom faces of the prism.

MAJRAD
Radius of the major (or circumscribed) circle of the polygon defining the top and bottom faces of the prism. Not used if *LSIDE* is input.

MINRAD
Radius of the minor (or inscribed circle) of the polygon defining the top and bottom faces of the prism. Not used if *LSIDE* or *MAJRAD* is input.

Notes

Defines a regular prism volume centered about the working plane origin. The prism must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) The top and bottom faces are polygonal areas that are parallel to the working plane but neither face need be coplanar with (i.e., "on") the working plane. The first keypoint defined for each face is at θ = 0°. See the **RPR4** and **PRISM** commands for other ways to create prisms.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Circumscr Rad
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Inscribed Rad
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Side Length

---

RPSD, *IR*, *IA*, *IB*, *ITYPE*, *DATUM*, *Name*

Computes response power spectral density (PSD).

POST26: Special Purpose

MP ME ST <> <> <> <> <> <> PP ED

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previous variable, the previous variable will be overwritten with this result.

IA, *IB*

Reference numbers of the two variables to be operated on. If only one, leave *IB* blank.

ITYPE

Defines the type of response PSD to be calculated:

0,1

Displacement (default).

2

Velocity.

3

Acceleration.

DATUM

Defines the reference with respect to which response PSD is to be calculated:

1

Absolute value.

2

Relative to base (default).

Name

Thirty-two character name identifying variable on listings and displays. Embedded blanks are compressed for output.

Notes

This command computes response power spectral density (PSD) for the variables referenced by the reference numbers *IA* and *IB*. The variable referred by *IR* will contain the response PSD. **File.PSD** must be available for the calculations to occur. Requires the **STORE**,PSD command to be issued first.

Menu Paths

Main Menu>TimeHist Postpro>Calc Resp PSD

RSFIT, *RSlab*, *Slab*, *Name*, *Rmod*, *Ytrans*, *Yval*, *Xfilt*, *CONF*

Fit a response surface for an output parameter in a solution set.

PROBABILISTIC: Response Surfaces

MP ME ST DY <> PR EM <> FL PP ED

RSlab

Response surface set label. Identifies the response surface results for later postprocessing. This label can be used as *Rlab* for postprocessing response surface results, and for evaluating the fit results with **RSPLOT** and **RSPRNT**. You must have separate identification labels for the solution sets and for the response surface sets, because you can perform a response surface fit based on Monte Carlo results. For this case, the original Monte Carlo results are identified by the solution set label provided in the **PDEXE** command and the results generated with the fitted response surfaces are identified by the response surface set label (*RSlab*) provided here. *RSlab* must not contain blanks. Maximum length of this field is 16 characters; if this field contains more than 16 characters, it will be truncated to 16 characters.

Slab

Solution set label that identifies the probabilistic analysis containing the results to be fitted. This solution set label is defined with the **PDEXE** command.

Name

Parameter name. The parameter must have been previously defined as a random output parameter with the **PDVAR** command. The result values of this parameter (stored in the solution set identified by *Slab*) are fitted as a function of all random input variables.

Rmod

Regression model to use for the evaluation of the response surface.

LIN
: Regression model with linear terms only.

QUAD
: Regression model with linear and pure quadratic terms (no cross-terms).

QUAX
: Regression model with linear and all quadratic terms (including cross-terms). (Default.)

Ytrans

Option for the transformation type applied to the output parameter identified with *Name*.

NONE
: The values of the output parameter *Name* are not transformed (default).

EXP
: The values of the output parameter *Name* are transformed according to Y* = exp(Y). The transformed values Y* are used for the fitting process.

LOGA
: The values of the output parameter *Name* are transformed according to $Y^* = \log_a(Y)$, where the basis a is specified by *Yval*. The transformed values Y* are used for the fitting process.

LOGN
: The values of the output parameter *Name* are transformed according to Y* = ln(Y). The transformed values Y* are used for the fitting process.

LOG10
The values of the output parameter *Name* are transformed according to $Y^* = \log_{10}(Y)$. The transformed values Y* are used for the fitting process.

SQRT
The values of the output parameter *Name* are transformed according to $Y^* = \sqrt{y}$. The transformed values Y* are used for the fitting process.

POW
The values of the output parameter *Name* are transformed according to $Y^* = Y^a$, where the exponent *a* is specified by *Yval*. The transformed values Y* are used for the fitting process.

BOX
The values of the output parameter *Name* are transformed according to the Box-Cox transformation.

$$Y^* = \frac{Y^{\lambda_i - 1}}{\lambda_i} \quad \text{for } \lambda_i \neq 0$$

$Y^* = \ln(Y)$ for $\lambda_i = 0$

The parameter λ_i is automatically determined within the interval [-2, 2]. The step length for this interval search is given in *Yval*. The transformed values Y* are used for the fitting process.

Yval
Value needed for *Ytrans* = LOGA, *Ytrans* = POW, and *Ytrans* = BOX. Ignored for all other options of *Ytrans*. For *Ytrans* = LOGA and *Ytrans* = POW, the default value is *Yval* = 1.0. For *Ytrans* = BOX, the default value is *Yval* = 0.01.

Xfilt
Option that specifies if irrelevant terms of the regression model should be filtered out.

NONE
Regression term filtering is not done. A full regression model as specified with the *Rmod* option is evaluated.

FSR
Use the "forward-stepwise-regression" to automatically filter out individual and irrelevant terms of the regression model. If irrelevant terms are sorted out this leads to a smaller number of coefficients to be determined in the regression analysis, which then leads to increased accuracy of the coefficients of the remaining terms in the regression model (default).

CONF
Confidence level that is used to filter the terms of the regression model if the *Xfilt* = FSR option has been selected. The value of *CONF* must be between 0.0 and 1.0 (default is 0.95). The higher this value, the more terms will be filtered out. Consequently, higher *CONF* values lead to a fewer terms in the regression model. Likewise, lower *CONF* values lead to more terms being included in the regression model.

Command Default

Rmod = QUAX, *Ytrans* = NONE, *Xfilt* = FSR, *CONF* = 0.95, *Yval* as explained above.

Notes

Evaluates a response surface for a random output parameter *Name*. The result values for the parameter *Name* as stored in the solution set *Slab* are fitted as a function of all random input parameters. There can be only one response surface set for each solution set. A response surface set can include the fitted response surfaces of one or more output parameters.

Note — If the **RSFIT** command is used for an existing response surface set for which Monte Carlo simulations have already been generated using the **RSSIMS** command then these Monte Carlo samples are deleted by the **RSFIT** command. In this case they need to be generated again using the **RSSIMS** command. It is necessary to delete the Monte Carlo simulations in order to make sure that the samples, their statistics and correlations are consistent with the response surfaces.

Menu Paths

Main Menu>Prob Design>Response Surf>Fit Resp Surf

RSPEED, *MIPS*, *SMFLOP*, *VMFLOP*

Supplies system performance information for use in time estimates.

RUNSTATS: Run Statistics Estimator

MP ME ST <> <> PR EM <> <> PP ED

MIPS
MIPS rating of computer (defaults to 4). This value is ignored if *SMFLOP* is specified.

SMFLOP
Scalar MFLOPS rating of computer (defaults to *MIPS*/4).

VMFLOP
Vector MFLOPS rating of computer (Defaults to *MIPS*/2).

Notes

Supplies system performance information to the program for its use in estimating run times [**RTIMST**]. Normally this command is invoked through the SETSPEED macro as executed by the ANSYS installation process.

Menu Paths

Main Menu>Run-Time Stats>System Settings

RSPLOT, *RSlab*, *YName*, *X1Name*, *X2Name*, *Type*, *NPTS*, *PLOW*, *PUP*

Plot a response surface.

PROBABILISTIC: Response Surfaces

MP ME ST DY <> PR EM <> FL PP ED

RSlab
Response Surface set label. Identifies the response surfaces generated by the **RSFIT** command.

YName
Parameter name. The parameter must have been previously defined as a random output parameter with the **PDVAR** command.

X1Name
Parameter name. The parameter must have been previously defined as a random input variable with the **PDVAR** command.

X2Name
Parameter name. The parameter must have been previously defined as a random input variable with the **PDVAR** command. *X2Name* must be different than *X1Name*.

Type
Type of the response surface visualization.

2D
2-D contour plot.

3D
3-D surface plot.

NPTS
Number of grid points for both the X1-axis and the X2-axis. The grid points are used for the evaluation of the response surface. The number must be between 1 and 500. Defaults to 20. If *NPTS* = 0 or greater than 500, then a value of 20 is used.

PLOW
Lower probability level used to determine the lower boundary (plotting range) of the curve in case the random input variable does not have a minimum value (such as Gauss). This probability must be between 0.0 and 1.0. Defaults to 0.0025.

PUP
Upper probability level used to determine the upper boundary of the curve. This probability must be between 0.0 and 1.0. Defaults to 0.9975.

Command Default

NPTS = 20, *PLOW* = 0.0025 (applied if distribution type has no minimum), *PUP* = 0.9975 (applied if distribution type has no maximum)

Notes

Plots the response surface of an output parameter *YName* as a function of two input parameters *X1Name* and *X2Name*.

If *PLOW* is left blank, then a minimum value of the distribution is used for plotting, provided it exists (for example, uniform distribution). If the distribution type has no minimum value (for example, Gaussian distribution), then the default value is used to determine the lower plotting range value. The same is true for the maximum value if *PUP* is left blank.

In addition to the response surface, the sampling points that are fitted by the response surface are also plotted by this command. However, sampling points falling outside of the plotting range defined by the *PLOW* and *PUP* fields will not be shown in the plot.

Menu Paths

Main Menu>Prob Design>Response Surf>Plt Resp Surf

RSPRNT, *RSlab*, *YName*, *Xout*

Print a response surface.

PROBABILISTIC: Response Surfaces

MP ME ST DY <> PR EM <> FL PP ED

RSlab

Response Surface set label. Identifies the response surfaces generated by the **RSFIT** command.

YName

Parameter name. The parameter must have been previously defined as a random output parameter with the **PDVAR** command. Identifies the parameter for which a response surface has been generated by the **RSFIT** command.

Xout

An option if an extended print-out of more feedback about goodness-of-fit and the details of the regression analysis of the response surface is requested.

No –

Use the standard print-out (default).

Yes –

Use the extended print-out.

Notes

Prints the results and details of a response surface analysis generated by the **RSFIT** command. For the specified output parameter *Yname*, the fitting details such as the individual terms of the response surface model and their corresponding coefficients are listed. The command also produces a comparison of the original values of *Yname* used for the fitting process and the approximate values derived from the fitting, and some goodness of fit measures.

If *Xout* = Yes, then more information about the regression analysis of the response surface will be printed. For example, the confidence intervals on the regression coefficients and the correlation between the regression coefficients among others.

Menu Paths

Main Menu>Prob Design>Response Surf>Prn Resp Surf

RSSIMS, *RSlab*, *NSIM*, *Seed*

Performs Monte Carlo simulations on response surface(s).

PROBABILISTIC: Response Surfaces

MP ME ST DY <> PR EM <> FL PP ED

RSlab

Response Surface set label. Identifies the response surfaces generated by the **RSFIT** command.

NSIM

Number of simulation loops on the response surfaces that will be generated for all random output parameters. If the **RSSIMS** command is issued multiple times using the same response surface set label the *NSIM* Monte Carlo simulations is appended to previous ones. The default value for *NSIM* is 10,000.

Seed

Seed value label. Random number generators require a seed value that is used to calculate the next random number. After each random number generation finishes, the seed value is updated and is used again to calculate the next random number. By default ANSYS initializes the seed value with the system time (one time only) when the ANSYS session started.

CONT

Continues updating using the derived seed value (default).

TIME

Initializes the seed value with the system time. You can use this if you want the seed value set to a specific value for one analysis and then you want to continue with a "random" seed in the next analysis. It is not recommended to "randomize" the seed value with the *Seed* = TIME option for multiple analyses. If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

INIT

Initializes the seed value using 123457. This value is a typical recommendation used very often in literature. This option leads to identical random numbers for all random input variables when the exact analysis will be repeated, making it useful for benchmarking and validation purposes (where identical random numbers are desired). If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

Value

Uses the specified (positive) value for the initialization of the seed value. This option has the same effect as *Seed* = INIT, except you can chose an arbitrary (positive) number for the initialization. If the Monte Carlo simulations requested with this command will be appended to previously existing simulations, then the *Seed* option is ignored and *Seed* = CONT is used.

Command Default

NSIM = 10,000, *Seed* = CONT

Notes

Generate the Monte Carlo simulations on the response surfaces that are included in a response surface set. Simulations are evaluated only for the output parameters that have been fitted in a response surface set using the **RSFIT** command.

If the **RSSIMS** command is issued multiple times using the same response surface label the probabilistic design system appends the samples generated here to the previous ones. This way you can start with a moderate *NSIM* number and add more samples if the probabilistic results are not accurate enough.

Menu Paths

Main Menu>Prob Design>Response Surf>RS Simulation

RSTAT

Prints the FE model statistics of the model.

RUNSTATS: Run Statistics Estimator

MP ME ST <> <> PR EM <> FL PP ED

Notes

Prints the finite element model statistics of the model currently in the database. The maximum node and element number used are displayed, as well as the number of nodes and elements selected.

Menu Paths

Main Menu>Run-Time Stats>Individual Stats

RSYS, *KCN*

Activates a coordinate system for printout or display of results.

POST1: Controls

MP ME ST <> <> PR EM <> FL PP ED

KCN

Coordinate system reference number. *KCN* may be 0,1,2 or any existing local coordinate system number. If *KCN* = SOLU, results are reported in whatever coordinate systems were associated with the results when calculated in the solution phase (i.e., the nodal and element coordinate systems). The exception is for layered shell and solid elements, for which data are transformed into the element coordinate system if **LAYER** = 0 or if no **LAYER** command is issued.

Command Default

Activate global Cartesian (*KCN* = 0) coordinate system.

Notes

Activates a coordinate system for the printout or display of results data. Results data will be rotated to this system during printout, display, or element table operations [**PRNSOL**, **PRESOL**, **PLNSOL**, **ETABLE**, etc.]. Coordinate systems can be defined with various commands [**LOCAL**, **CS**, **CLOCAL**, **CSKP**, ...]. If **RSYS** is issued with *KCN* > 10 (i.e., a local coordinate system), and the specified system is subsequently redefined, you must reissue **RSYS** for results to be rotated into the redefined system.

Rotated nodal data are any of the items shown for the **PRNSOL** command having the *Comp* label COMP, such as U (displacements), S (stresses), etc. Nodal results can be properly rotated only if the resulting component set is consistent with the degree-of-freedom set at the node (the degree-of-freedom set at a node is determined by the elements attached to the node). For example, if a node does not have a UZ degree of freedom during solution, then any Z component resulting from a rotation will not print or display in POST1. Therefore, results at nodes with a single degree-of-freedom (UY only, for example) should not be rotated; that is, they should be viewed only in the nodal coordinate system or a system parallel to the nodal system (Note that the command default is the global Cartesian system, which may not be parallel to the nodal system). Results at nodes with a 2-D degree-of-freedom set (UX and UY, for example) should not be rotated out of the 2-D plane.

Element component results in the database from the solution phase (*KCN* = SOLU) are in the element coordinate systems. For nearly all the solid elements, the default element coordinate systems are parallel to the global Cartesian coordinate system. For the shell elements and the remaining solid elements, the default element coordinate system can differ from element to element. For layered shell and layered solid elements, the default coordinate system can vary from layer to layer within the element. The element coordinate system is initially defined with the **ESYS** command.

If large deflection is active, the element component result directions are rotated by the amount of rigid body rotation (does not apply to the 18x elements). However, the hyperelastic elements always produce stresses and strains in the specified results coordinate system; no rigid body rotation is added for HYPER56, HYPER58, HYPER74, HYPER84, HYPER86, and HYPER158. This is also true for elements that contain hyperelastic material properties (**TB**,HYPER), such as, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SHELL208, and SHELL209. Component results displayed in the co-rotated coordinate system include the element rigid body rotation from the initial global coordinate system (except for the elements listed above). All other element result transformations are relative to the rotated global system. For the listed elements, the element component results are displayed in the initial global coordinate system.;All other element result transformations are therefore also relative to the initial global system. Nodal degree-of-freedom results are based on the initial (and not the updated) geometry.

PowerGraphics does not support **RSYS**,SOLU.

When you generate a PGR file in SOLUTION, you can use the Results Viewer to display your stresses only in the coordinate system in which you write your PGR file. If you wish to view stresses in other coordinate systems, you can load your results file into the Results Viewer and regenerate the data. See Using the PGR File in POST1 for more information on using the Results Viewer.

RSYS has no effect on beam stresses (BEAM3, BEAM4, PIPE16, PIPE20, BEAM23, BEAM24, BEAM44, BEAM54, PIPE60, BEAM188, and BEAM189), which are always displayed (with **/ESHAPE**,1 and PowerGraphics) in the element coordinate system.

For PowerGraphics, **PLVECT** arrow displays (temperature, velocity, force, etc.) will be plotted in **RSYS** = 0. Subsequent operations will revert to your original coordinate system.

RSYS can be used to rotate stress data for all explicit (ANSYS LS-DYNA) elements except BEAM161, COMBI165, and composite SHELL163 (KEYOPT(3) = 1). In models that contain these element types combined with other explicit elements, you must unselect the unsupported elements before issuing the **RSYS** command. **RSYS** does not support strain data for any explicit element types. If you request strain results for explicit elements when **RSYS** is not set to the global Cartesian coordinate system (*KCN* = 0), the printing or plotting command will be ignored. (Displacements are always rotated into the results coordinate system, independent of the explicit element type.)

Menu Paths

Main Menu>General Postproc>Options for Outp
Utility Menu>List>Results>Options

RTHICK, *Par*, *ILOC*, *JLOC*, *KLOC*, *LLOC*

Defines variable thickness at nodes for shell elements.

PREP7: Meshing

MP ME ST DY <> PR <> <> <> PP ED

Par
Array parameter (indexed by node number) that expresses the function to be mapped. For example, func (17) should be the desired shell thickness at node 17.

ILOC
Position in real constant set for thickness at node I of the element (default 1).

JLOC
Position in real constant set for thickness at node J of the element (default 2).

KLOC
Position in real constant set for thickness at node K of the element (default 3).

LLOC
Position in real constant set for thickness at node L of the element (default 4).

Notes

After **RTHICK**, each selected element will have its REAL number matching its ELEM number, for example, R (*ILOC*) = func (I NODE), R (*JLOC*) = func (J NODE), etc. Any other real constants on a previously defined real constant set will remain unchanged. This command cannot be used for beam elements.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Real Constants>Thickness Func
Main Menu>Preprocessor>Real Constants>Thickness Func
Main Menu>Solution>Load Step Opts>Other>Real Constants>Thickness Func

---

RTIMST

Prints runtime estimates.

RUNSTATS: Run Statistics Estimator

MP ME ST <> <> PR EM <> <> PP ED

Notes

Prints runtime estimates for the current model. A runtime estimate is given for the first iteration, for subsequent iterations, and for total runtime. The runtime estimates are itemized for such ANSYS solution components as element formulation, wavefront solution, back substitution, etc. The estimates will be for a computer system

using performance attributes as described by the **RSPEED** command and an estimated number of iterations as specified on the **RITER** command. The **RSPEED** and **RITER** commands should be input before this command.

Note — This command will cause reordering [**WAVES**] if reordering was not already done on the model.

Menu Paths

Main Menu>Run-Time Stats>Individual Stats

RUN, *DX*, *DY*, *DZ*, *NDIV*, *NEND*, *ESTRT*, *EINC*

Defines a pipe run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

DX, *DY*, *DZ*
Increment (in terms of the active coordinate system components) to determine run end point. Increment is applied to branch starting point [**BRANCH**] or end point of previous run (whichever was later).

NDIV
Number of divisions (elements) along branch (defaults to 1). A node is generated at the end of each division.

NEND
Number to be assigned to end node of branch (defaults to MAXNP + *NDIV*).

ESTRT
Number to be assigned to first element of branch (defaults to the previous maximum element number (MAXEL) + 1).

EINC
Element number increment (defaults to 1).

Notes

Defines a pipe run from a previous point to an incremental point. Nodes (and elements) are generated straight (in the active coordinate system). Elements are of type PIPE16 straight pipes. Material properties, real constants, and loads are derived from the previously defined piping specifications. Piping loads and specifications are defined with the **PCORRO**, **PDRAG**, **PFLUID**, **PINSUL**, **POPT**, **PPRES**, **PSPEC**, **PTEMP**, and **PUNIT** commands. Generated items may be listed (or displayed) with the standard commands (**NLIST**, **ELIST**, **NPLOT**, **EPLOT**, **ETLIST**, **RLIST**, etc.). Items may also be modified (**NMODIF**, **EMODIF**, **RMODIF**, etc.) or redefined as desired. See the *ANSYS Modeling and Meshing Guide* for details.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Define Pipes>Pipe Run

---

/RUNST

Enters the run statistics processor.

SESSION: Processor Entry
RUNSTATS: Run Statistics Estimator

MP ME ST <> <> PR EM <> FL PP ED

Notes

Enters the run statistics processor for estimating various solution size and speed statistics. Statistics include run time estimates, wavefront estimates, file size estimates, memory requirements and finite element model size information.

This command is valid only at the Begin Level.

Menu Paths

Main Menu>Run-Time Stats

RWFRNT

Generates wavefront statistics and memory requirements.

RUNSTATS: Run Statistics Estimator

MP ME ST <> <> PR EM <> <> PP ED

Notes

Generates current solver statistics, such as maximum and RMS wavefront, and memory requirements based on the chosen solver (**EQSLV** command). The **RWFRNT** command causes reordering if reordering was not already done on the model.

Menu Paths

Main Menu>Run-Time Stats>Individual Stats

S Commands

SABS, *KEY*

Specifies absolute values for element table operations.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

KEY
Absolute value key:

0
Use algebraic values in operations.

1
Use absolute values in operations.

Command Default

Use algebraic values.

Notes

Causes absolute values to be used in the **SADD**, **SMULT**, **SMAX**, **SMIN**, and **SSUM** operations.

Menu Paths

Main Menu>General Postproc>Element Table>Abs Value Option

SADD, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*, *CONST*

Forms an element table item by adding two existing items.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

LabR
Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

Lab1
First labeled result item in operation.

Lab2
Second labeled result item in operation (may be blank).

FACT1
Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

FACT2
Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

CONST
Constant value.

Notes

Forms a labeled result (see **ETABLE** command) for the selected elements by adding two existing labeled result items according to the operation:

LabR = (*FACT1* x *Lab1*) + (*FACT2* x *Lab2*) + *CONST*

May also be used to scale results for a single labeled result item. If absolute values are requested [**SABS**,1], absolute values of *Lab1* and *Lab2* are used.

Menu Paths

Main Menu>General Postproc>Element Table>Add Items

SALLOW, *STRS1*, *STRS2*, *STRS3*, *STRS4*, *STRS5*, *STRS6*

Defines the allowable stress table for safety factor calculations.

POST1: Element Table

MP ME ST DY <> PR <> <> <> PP ED

STRS1, *STRS2*, *STRS3*, *STRS4*, *STRS5*, *STRS6*

Input up to six allowable stresses corresponding to the temperature points [**TALLOW**].

Notes

Defines the allowable stress table for safety factor calculations [**SFACT**,**SFCALC**]. Use the **STAT** command to list current allowable stress table. Repeat **SALLOW** to zero table and redefine points (6 maximum).

Safety factor calculations are not supported by PowerGraphics. Both the **SALLOW** and **TALLOW** commands must be used with the Full Model Graphics display method active.

Menu Paths

Main Menu>General Postproc>Safety Factor>Allowable Strs>Constant
Main Menu>General Postproc>Safety Factor>Allowable Strs>Reset Stress
Main Menu>General Postproc>Safety Factor>Allowable Strs>Temp-depend

---

SARPLOT, *Prefer*, *VALUE*

Displays areas smaller than a specified size (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

Prefer

Preference for area display. If *Prefer* = FACTOR, the command displays all areas whose size is smaller than the size of the average area within the model times *VALUE*. This is the default preference. If *Prefer* = AREA, the command displays all areas that are smaller than that specified by *VALUE*. If *Prefer* = NARROW, the command displays all areas that have an aspect ratio greater than *VALUE* (useful for finding "sliver" areas).

VALUE
Numeric value used as argument for *Prefer*.

Notes

Use this command to locate and display disproportionately small areas when repairing the geometry of models imported from CAD files. Areas matching the criteria specified in *Prefer* and *VALUE* both display in a different color and include their IDs. This command is available only for models imported through the Default IGES option.

Menu Paths

Main Menu>Preprocessor>Modeling>Simplify>Detect/Display>Small Areas

---

SAVE, *Fname*, *Ext*, --, *Slab*
Saves all current database information.

DATABASE: Set Up

MP ME ST DY <> PR EM EH FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to DB if *Fname* is blank.

--
Unused field

Slab
Mode for saving the database:

ALL
Save the model data, solution data and post data (element tables, etc.). This value is the default.

MODEL
Save the model data (solid model, finite element model, loadings, etc.) only.

SOLU
Save the model data and the solution data (nodal and element results).

Notes

Saves all current database information to a file (**File.DB**). In interactive mode, an existing **File.DB** is first written to a backup file (**File.DBB**). In batch mode, an existing **File.DB** is replaced by the current database information with no backup. The command should be issued periodically to ensure a current file backup in case of a system "crash" or a "line drop." It may also be issued before a "doubtful" command so that if the result is not what was intended the database may be easily restored to the previous state. A save may be time consuming for large models. Repeated use of this command overwrites the previous data on the file (but a backup file is first written

during an interactive run). When issued from within POST1, the nodal boundary conditions in the database (which were read from the results file) will overwrite the nodal boundary conditions existing on the database file.

This command is valid in any processor.

Menu Paths

Utility Menu>File>Save as
Utility Menu>File>Save as Jobname.db

SBCLIST

Lists solid model boundary conditions.

SOLUTION: Misc Loads

MP ME ST <> <> PR EM <> <> PP ED

Notes

Lists all solid model boundary conditions for the selected solid model entities. See also **DKLIST**, **DLLIST**, **DALIST**, **FKLIST**, **SFLLIST**, **SFALIST**, **BFLLIST**, **BFALIST**, **BFVLIST**, and **BFKLIST** to list items separately.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Solid Model Loads

SBCTRAN

Transfers solid model loads and boundary conditions to the FE model.

SOLUTION: Misc Loads

MP ME ST <> <> PR EM <> <> PP ED

Notes

Causes a manual transfer of solid model loads and boundary conditions to the finite element model. Loads and boundary conditions on unselected keypoints, lines, areas, and volumes are not transferred. Boundary conditions and loads will not be transferred to unselected nodes or elements. The **SBCTRAN** operation is also automatically done upon initiation of the solution calculations [**SOLVE**].

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>All Solid Lds
Main Menu>Solution>Define Loads>Operate>Transfer to FE>All Solid Lds

---

SDELETE, *SFIRST*, *SLAST*, *SINC*, *KNOCLEAN*

Deletes cross sections from the ANSYS database.

PREP7: Cross Sections

MP ME ST <> <> <> <> <> <> PP ED

SFIRST
First section ID to be deleted; defaults to first available section in the database.

SLAST
Last section ID to be deleted; defaults to last available section in the database.

SINC
Increment of the section ID; defaults to 1.

KNOCLEAN
Pretension element cleanup key (pretension sections only).

0
Perform cleanup of pretension elements (delete pretension elements and reconnect elements split during PSMESH).

1
Do not perform cleanup.

Notes

Deletes one or more specified sections and their associated data from the ANSYS database.

Menu Paths

Main Menu>Preprocessor>Sections>Delete Section

SE, *File*, --, --, *TOLER*

Defines a superelement.

PREP7: Superelements

MP ME ST <> <> PR EM EH <> PP ED

File
Jobname (8 character maximum) of file containing superelement. Defaults to the current **Jobname**.

--, --
Unused fields.

TOLER
Tolerance used to determine if use pass nodes are noncoincident with master nodes having the same node numbers. Defaults to 0.0001. Use pass nodes will always be replaced by master nodes of the same node number. However, if a use pass node is more than *TOLER* away from the corresponding master node, a warning is generated.

Notes

Defines a superelement by reading in the superelement matrices and master nodes from the superelement matrix file. The matrix file (**File.SUB**) must be available from the substructure generation pass. The proper element type (MATRIX50) must be active [**TYPE**] for this command. A scratch file called **File.SORD** showing the superelement names and their corresponding element numbers is also written.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>From .SUB File

---

SECCONTROLS, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*

Overrides program calculated properties.

PREP7: Cross Sections

MP ME ST <> <> PR <> <> <> PP ED

VAL1, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*

Values, such as the length of a side or the numbers of cells along the width, that describe the geometry of a section. See the Notes section of this command description for details about these values for the various section types.

Notes

The **SECCONTROLS** command is divided into two types: BEAMS and SHELLS. The beam type is listed first, followed by the shell type. Values are associated with the most recently issued **SECTYPE** command. The data required is determined by the section type, and is different for each type.

SECCONTROLS overrides the program calculated transverse shear stiffness.

The **SECCONTROLS** command does not apply to the thermal shell elements, SHELL131 and SHELL132.

BEAMS

Type: BEAM

Data to be supplied in the value fields:
TXZ, - , *TXY*, *ADDMAS*
TXZ = User transverse shear stiffness.
- = Unused field.
TXY = User transverse shear stiffness.
ADDMAS = Added mass per unit length.

SHELLS

Type: SHELL

Data to be supplied in the value fields:
E_{11}, E_{22}, E_{12}, *ADDMAS*, *MEMSCF*, *BENSCF*, *DRLSTIF*

E_{11} = User transverse shear stiffness.
E_{22} = User transverse shear stiffness.
E_{12} = User transverse shear stiffness.
ADDMAS = Added mass per unit area.
MEMSCF = Hourglass control membrane scale factor.
BENSCF = Hourglass control bending scale factor.
DRLSTIF = Drill stiffness scale factor.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Sect Control
Main Menu>Preprocessor>Sections>Shell>Add / Edit

SECDATA, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*, *VAL9*, *VAL10*

Describes the geometry of a section.

PREP7: Cross Sections

MP ME ST <> <> PR <> <> <> PP ED

VAL1, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*, *VAL9*, *VAL10*
Values, such as thickness or the length of a side or the numbers of cells along the width, that describe the geometry of a section. The terms *VAL1*, *VAL2*, etc. are specialized for each type of cross-section. See the **Notes** section of this command description for details about these values for the various section types.

Notes

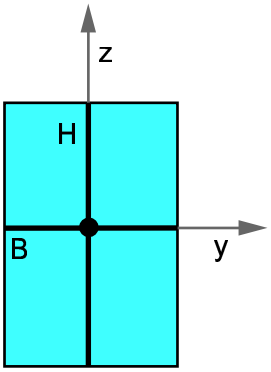
The **SECDATA** command is divided into the types: BEAMS, TAPER, SHELLS, PRETENSION, and JOINTS. The beam types are collectively listed first, followed by the remaining section types.

SECDATA defines the data describing the geometry of a section. The data input on the **SECDATA** command is interpreted based on the most recently issued **SECTYPE** command. The data required is determined by the section type and subtype, and is different for each one.

BEAMS

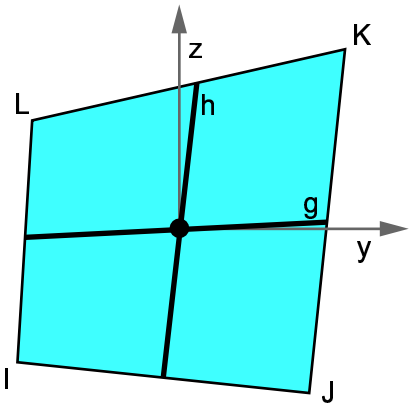
Beam sections are referenced by the BEAM44, BEAM188, or BEAM189 elements. A section made of multiple materials cannot be modeled using BEAM44. Not all **SECOFFSET** location values are valid for each subtype.

Type: BEAM, Subtype: RECT



Data to be supplied in the value fields:
B, *H*, *Nb*, *Nh*
B = Width
H = Height
Nb = Number of cells along width; default = 2
Nh = Number of cells along height; default = 2
*Nb***Nh*

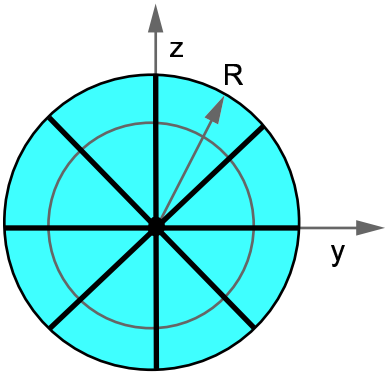
Type: BEAM, Subtype: QUAD



Data to be supplied in the value fields:
yI, *zI*, *yJ*, *zJ*, *yK*, *zK*, *yL*, *zL*, *Ng*, *Nh*
yI, *zI*, *yJ*, *zJ*, *yK*, *zK*, *yL*, *zL* = Coordinate location of various points
Ng = Number of cells along *g*; default = 2
Nh = Number of cells along *h*; default = 2
*Ng***Nh*

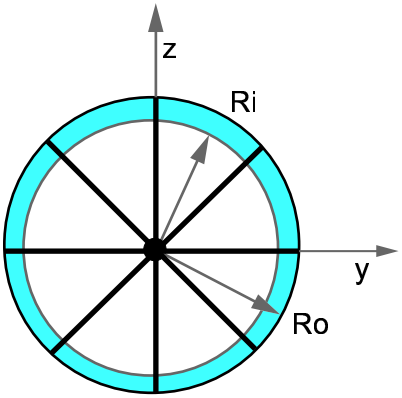
Note — Degeneration to triangle is permitted by specifying the same coordinates for cells along an edge.

Type: BEAM, Subtype: CSOLID



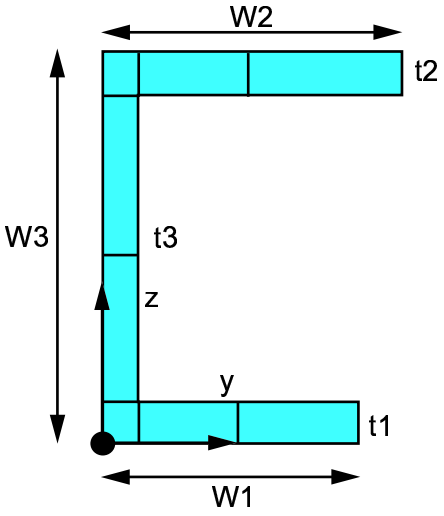
Data to be supplied in the value fields:
R, *N*, *T*
R = Radius
N = Number of divisions around the circumference.
T = Number of divisions through the radius

Type: BEAM, Subtype: CTUBE



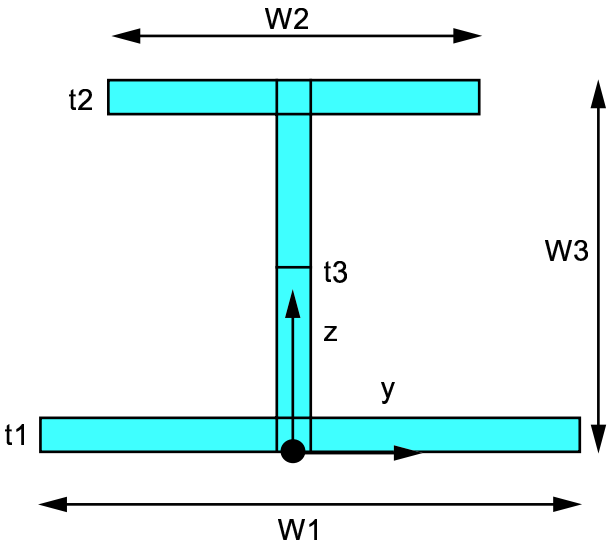
Data to be supplied in the value fields:
Ri, *Ro*, *N*
Ri = Inner radius of the tube
Ro = Outer radius of the tube
N = Number of cells along the circumference; default = 8

Type: BEAM, Subtype: CHAN



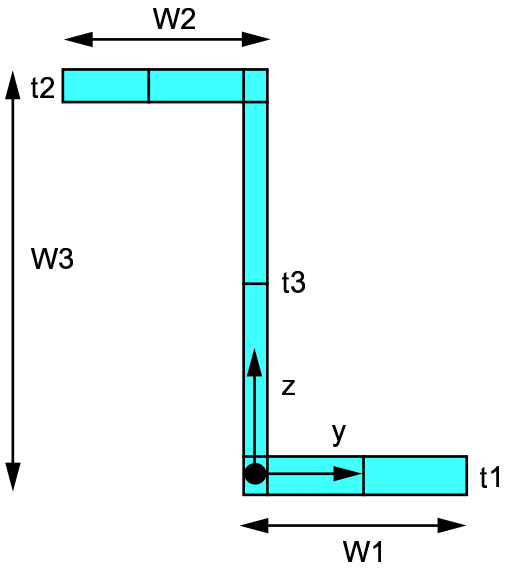
Data to be supplied in the value fields:
W1, *W2*, *W3*, *t1*, *t2*, *t3*
W1, *W2* = Lengths of the flanges
W3 = Overall depth
t1, *t2* = Flange thicknesses
t3 = Web thicknesses

Type: BEAM, Subtype: I



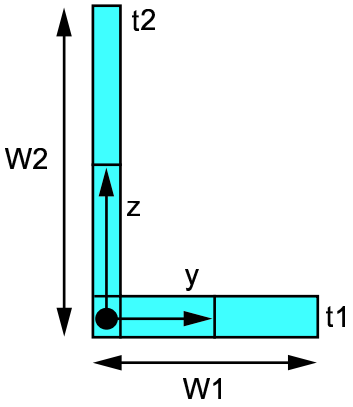
Data to be supplied in the value fields:
W1, *W2*, *W3*, *t1*, *t2*, *t3*
W1, *W2* = Width of the top and bottom flanges
W3 = Overall depth
t1, *t2* = Flange thicknesses
t3 = Web thicknesses

Type: BEAM, Subtype: Z



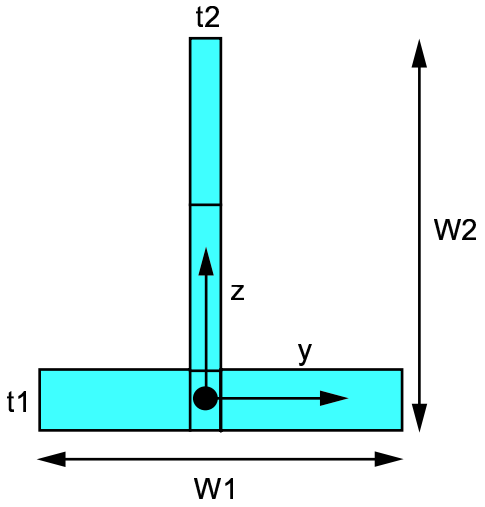
Data to be supplied in the value fields:
W1, *W2*, *W3*, *t1*, *t2*, *t3*
W1, *W2* = Flange lengths
W3 = Overall depth
t1, *t2* = Flange thicknesses
t3 = Stem thicknesses

Type: BEAM, Subtype: L



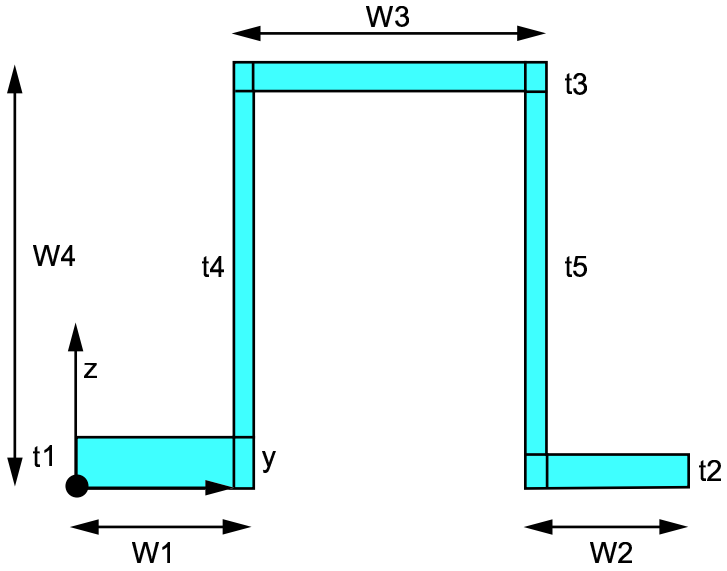
Data to be supplied in the value fields:
W1, *W2*, *t1*, *t2*
W1, *W2* = Leg lengths
t1, *t2* = Leg thicknesses

Type: BEAM, Subtype: T



Data to be supplied in the value fields:
W1, *W2*, *t1*, *t2*
W1 = Flange width
W2 = Overall depth
t1 = Flange thicknesses
t2 = Stem thicknesses

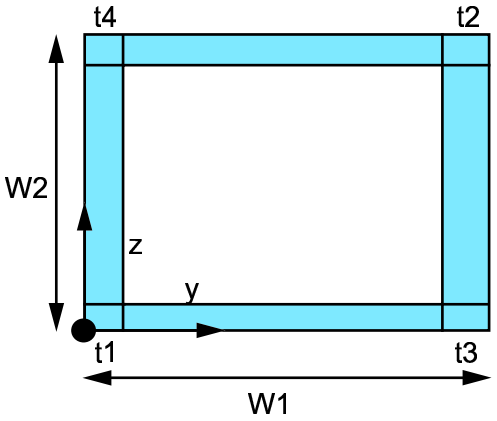
Type: BEAM, Subtype: HATS



Data to be supplied in the value fields:
W1, *W2*, *W3*, *W4*, *t1*, *t2*, *t3*, *t4*, *t5*
W1, *W2* = Width of the brim
W3 = Width of the top of the hat
W4 = Overall depth
t1, *t2* = Thickness of the brim
t3 = Thickness of the top of the hat

t4, *t5* = Web thicknesses

Type: BEAM, Subtype: HREC



Data to be supplied in the value fields:
W1, *W2*, *t1*, *t2*, *t3*, *t4*
W1 = Outer width of the box
W2 = Outer height of the box
t1, *t2*, *t3*, *t4* = Wall thickness

Type: BEAM, Subtype: ASEC

Arbitrary: User-supplied integrated section properties instead of basic geometry data.
Data to be supplied in the value fields:
A, *Iyy*, *Iyz*, *Izz*, *Iw*, *J*, *CGy*, *CGz*, *SHy*, *SHz*
A = Area of section
Iyy = Moment of inertia about the y axis
Iyz = Product of inertia
Izz = Moment of inertia about the z axis
Iw = Warping constant
J = Torsional constant
CGy = Y coordinate of centroid
CGz = Z coordinate of centroid
SHy = Y coordinate of shear center
SHz = Z coordinate of shear center
SECPLOT cannot display an ASEC plot.

Type: BEAM, Subtype: MESH

User-defined mesh.
Data required is created by the **SECWRITE** command and is read into ANSYS by the **SECREAD** command. See the **SECREAD** command for more information about this data.
When user mesh is input using **SECREAD**, ANSYS calculates the area, second moments of inertia, centroid, torsion constant, warping rigidity, and shear center.

If you redefine a material for a composite cross-section after creating the section, you must reissue the **SECREAD** command.

TAPER

Type: TAPER

Taper sections are referenced by BEAM188, and BEAM189 elements. Issue this command two times (once for each end of the tapered beam) following the **SECTYPE**,,TAPER command.

Data to be supplied in the value fields:
Sec_IDn, *XLOC*, *YLOC*, *ZLOC*
Sec_IDn = Previously defined beam section at ends 1 (I) and 2 (J).
XLOC, *YLOC*, *ZLOC* = The location of Sec_ID*n* in global space.

SHELLS

Type: SHELL

Shell sections are referenced by the SHELL131, SHELL132, SHELL181, SHELL208, and SHELL209 elements.

Data to be supplied in the value fields:
TK, *MAT*, *THETA*, *NUMPT*
TK = Thickness of shell layer. Thickness may be tapered using the **SECFUNCTION** command. Use zero for a ply drop-off for SHELL181, SHELL208, and SHELL209. Zero thickness is not allowed for SHELL131 and SHELL132. The sum of all layer thicknesses must be greater than zero.
MAT = Material ID for layer (any 18x material model is available for SHELL181, SHELL208, and SHELL209, including UserMat). MAT is required for a composite (multi-layered) laminate, (i.e., no default). For a homogeneous (single-layered) shell, the default is the MAT command setting. The TREF command is supported for SHELL181, SHELL208, and SHELL209, but not MP,REFT for each layer.
THETA = Angle (in degrees) of layer element coordinate system with respect to element coordinate system (ESYS).
NUMPT = Number of integration points in layer. The GUI permits 1, 3, 5, 7, or 9 points (default = 3). However a higher odd number may be specified in the command. The integration rule used is Simpson's Rule. (*NUMPT* is not used by SHELL131 and SHELL132.)

Use the **SECDATA** command for each layer in the section.

PRETENSION

Type: PRETENSION

Pretension sections are referenced by the PRETS179 element.

Data to be supplied in the value fields:
node, *nx*, *ny*, *nz*
node = Pretension node number.

nx = Orientation in global Cartesian x direction.
ny = Orientation in global Cartesian y direction.
nz = Orientation in global Cartesian z direction.

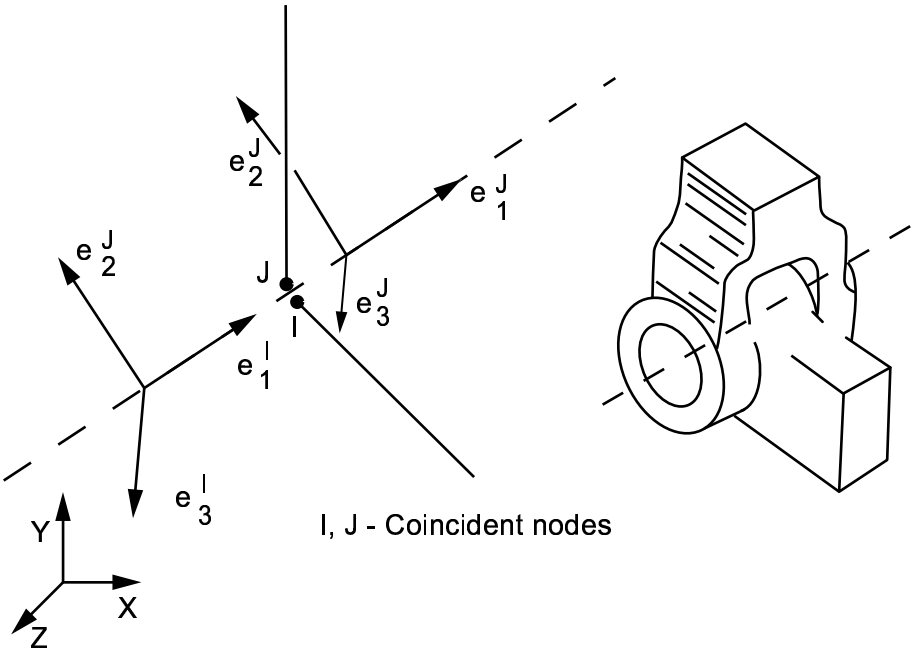
The following usage is typical:

```
SECTYPE, 1,PRETENSION
SECDATA, 13184, 0.000, 0.000, 1.000
SECMODIF, 1,NAME,example
SLOAD, 1,9, TINY,F, 100.00 , 1, 2
```

The *PRETENSION* section options of **SECTYPE** and **SECDATA** are documented mainly to aid in the understanding of data written by **CDWRITE**. We advise that you generate pretension sections using **PSMESH**.

JOINTS

Type: JOINT, Subtype: REVO

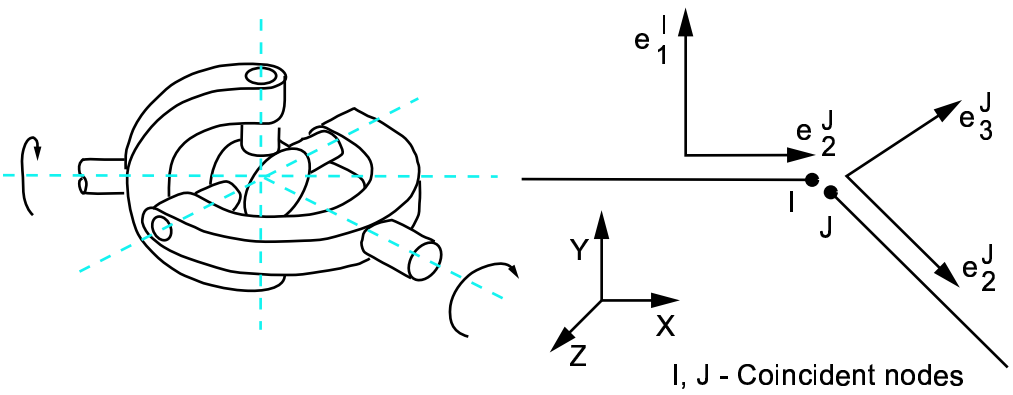


Data to be supplied in the value fields (VAL3 = angle1):
,,,angle1,,
angle1 refers to the component of rotation about the revolute axis.

Note — If the reference angle is not specified, it is calculated from the default or starting configuration of the revolute joint element. The reference angle is used in the constitutive calculations.

Type: JOINT, Subtype: UNIV

MPC184 Universal Joint Geometry



Data to be supplied in the value fields (VAL3 = angle1 and VAL5 = angle3):
,,,angle1,,angle3
angle1 and *angle3* refer to the rotational components of relative motion in the joint.

Note — Only two of the three vector components will show an angular displacement. The rotational components can therefore be expressed with two angles. If the reference angles are not specified, they are calculated from the default or starting configuration of the universal joint element. The reference angles are used in the constitutive calculations. See MPC184 for more intormation on joint elements.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Common Sections
Main Menu>Preprocessor>Sections>Beam>Taper Sections>By Picked Nodes
Main Menu>Preprocessor>Sections>Beam>Taper Sections>By XYZ Location
Main Menu>Preprocessor>Sections>Joints>Add / Edit
Main Menu>Preprocessor>Sections>Shell>Add / Edit

SECFUNCTION, *TABLE*

Specifies shell section thickness as a tabular function.

PREP7: Cross Sections

MP ME ST <> <> PR <> <> <> PP ED

TABLE
Table name reference for specifying tabular thickness as a function of global XYZ coordinates. To specify a table, enclose the table name in percent signs (%), e.g. **SECFUNCTION**,%tabname%). Use the ***DIM** command to define a table.

Notes

The table defined by the **SECFUNCTION** command is associated with the section most recently defined using the **SECTYPE** command.

The table defines your total shell thickness at any point in space. In multi-layered sections, the total thickness and each layer thickness are scaled accordingly.

The Function Builder is a convenient way to define your thickness tables. (**Utility Menu> Parameters> Functions> Define/Edit**)

To learn more about the Function Builder see Using the Function Editor of the *ANSYS Basic Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Sections>Shell>Add / Edit

SECJOINT, --, *ID1*, *ID2*

Defines local coordinate systems at the nodes that form the MPC184 joint element.

PREP7: Cross Sections

MP ME ST <> <> PR <> <> <> PP ED

`--`
Currently unused.

`ID1, ID2`
Identifiers of the local coordinate systems.

Notes

Issue another **SECJOINT** command to overwrite current values.

Menu Paths

Main Menu>Preprocessor>Sections>Joints>Add / Edit

/SECLIB, *Option*, *Path*

Sets the default section library path for the SECREAD command.

PREP7: Cross Sections

MP ME ST <> <> <> <> <> <> PP ED

`Option`

READ
Sets the read path (default).

STATUS
Reports the current section library path setting to the **Jobname.LOG** file.

Path
Defines the directory path from which to read section library files.

Notes

When the **SECREAD** command is issued without a directory path, the command searches for a section library in the following order:

- The user's home directory
- The current working directory
- The path specified by the **/SECLIB** command

Menu Paths

Main Menu>Preprocessor>Sections>Section Library>Library Path

SECLOCK, *dof*, *MINVALUE*, *MAXVALUE*, *dof*, *MINVALUE*, *MAXVALUE*, *dof*, *MINVALUE*, *MAXVALUE*

Specifies locks on the components of relative motion in a joint element.

PREP7: Cross Sections

MP ME ST <> <> PR <> <> <> PP ED

dof
Local degree of freedom to be locked.

MINVALUE
Low end of the range of allowed movement for the specified DOF.

MAXVALUE
High end of the range of allowed movement for the specified DOF.

Notes

Specify up to three DOFs to be locked. Locks are activated when the limit values are reached, and further motion in that DOF is frozen. If necessary, you may repeat the command.

Menu Paths

Main Menu>Preprocessor>Sections>Joints>Add / Edit

SECMODIF, *SECID*, *Kywrd*

Modifies a pretension section

PREP7: Cross Sections

MP ME ST <> <> <> <> <> <> PP ED

If *Kywrd* = NORM, command format is **SECMODIF**,*SECID*, NORM, *NX*, *NY*, *NZ*, *KCN*

SECID
Unique section number. This number must already be assigned to a section.

NORM
Keyword specifying that the command will modify the pretension section normal direction.

NX, *NY*, *NZ*
Specifies the individual normal components to modify.

KCN
Coordinate system number. This can be either 0 (Global Cartesian), 1 (Global Cylindrical) 2 (Global Spherical), 4 (Working Plane), 5 (Global Y Axis Cylindrical) or an arbitrary reference number assigned to a coordinate system.

If *Kywrd* = NAME, the command format is **SECMODIF**,*SECID*, NAME, *Name*

SECID
Unique section number. This number must already be assigned to a section.

NAME
Action key that instructs the command to change the name of the specified pretension section.

Name
The new name to be assigned to the pretension section.

Notes

The **SECMODIF** command either modifies the normal for a specified pretension section, or changes the name of the specified pretension surface.

Menu Paths

Main Menu>Preprocessor>Sections>Pretension>Modify Name
Main Menu>Preprocessor>Sections>Pretension>Modify Normal

SECNUM, *SECID*

Sets the element section attribute pointer.

PREP7: Cross Sections

MP ME ST <> <> PR <> <> <> PP ED

SECID
Defines the section ID number to be assigned to the subsequently-defined elements by the **LMESH**, **E**, or **EN** commands. Defaults to 1. See **SECTYPE** for more information about the section ID number.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs
Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

---

SECOFFSET, *Location*, *OFFSET1*, *OFFSET2*, *CG-Y*, *CG-Z*, *SH-Y*, *SH-Z*

Defines the section offset for cross sections.

PREP7: Cross Sections

MP ME ST <> <> PR <> <> <> PP ED

Location, *OFFSET1*, *OFFSET2*, *CG-Y*, *CG-Z*, *SH-Y*, *SH-Z*
The location of the nodes in the section. All are dependent on the type. See the Notes section of this command description for details about these values for the various section types.

Notes

The **SECOFFSET** command is divided into two types: BEAMS and SHELLS. The beam types are collectively listed first, followed by the shell types.

The offsets defined by the **SECOFFSET** command are associated with the section most recently defined using the **SECTYPE** command. Not all **SECOFFSET** location values are valid for each subtype.

For the thermal shell elements, SHELL131 and SHELL132, the node offset specified by **SECOFFSET** is used in thermal contact analyses. Otherwise, the **SECOFFSET** command has no affect on the solution for these elements and is used only for visualization purposes.

BEAMS

Type: BEAM

Data to be supplied in the value fields:

Location, *OFFSETY*, *OFFSETZ*, *CG-Y*, *CG-Z*, *SH-Y*, *SH-Z*

Location

CENT --
Beam node will be offset to centroid (default).

SHRC --
Beam node will be offset to shear center.

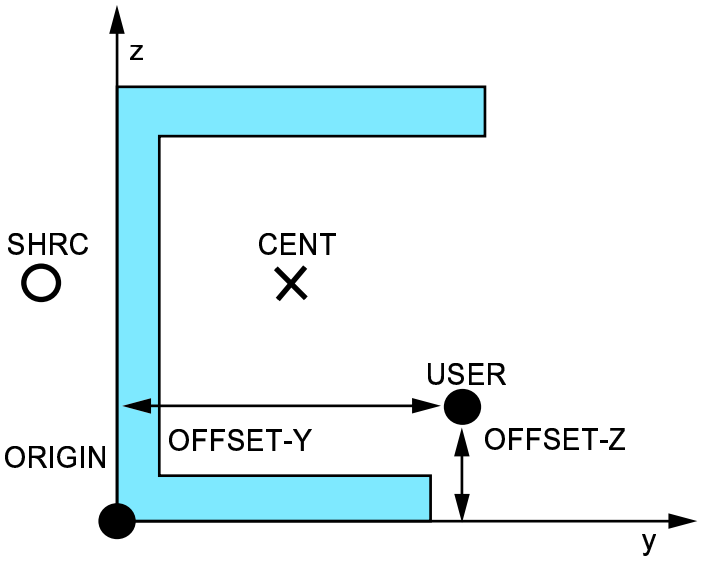
ORIGIN --
Beam node will be offset to origin of the cross section.

USER --
Beam node will be offset to the location specified by the *OFFSETY* and *OFFSETZ* arguments.

OFFSETY, *OFFSETZ*
Values that locate the node with respect to the default origin of the cross section when the *Location* argument is set to USER. Valid only when USER is set.

The following figure illustrates the offsets for a channel cross section, and shows the relative locations of SHRC and CENT.

Offsets for a CHAN Section Subtype



CG-Y, *CG-Z*, *SH-Y*, *SH-Z*
Override the program-calculated centroid and shear centroid locations.

Note — This option should only be used by *advanced users* modeling composite cross sections.

SHELLS

Type: SHELL

Data to be supplied in the value fields:

Location, *OFFSET*

Location

TOP --
Shell node will be offset to top of the section.

MID --
Shell node will be offset to midplane of the section (default).

BOT --
Shell node will be offset to bottom of the section.

USER --
Shell node will be offset to the location specified by the *OFFSET* argument.

OFFSET
Value that locates the node with respect to the default origin (midplane) of the section when the *Location* argument is set to USER. Valid only when USER is set.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Common Sections
Main Menu>Preprocessor>Sections>Beam>Custom Sections>Read Sect Mesh

Main Menu>Preprocessor>Sections>Shell>Add / Edit

SECPLOT, *SECID*, *VAL1*, *VAL2*

Plots the geometry of a beam or shell section to scale.

PREP7: Cross Sections

MP ME ST <> <> <> <> <> <> PP ED

SECID
The section ID number as defined by the **SECTYPE** command.

VAL1, *VAL2*
Values that control the information to be plotted. See the Notes section of this command description for details about these values for the various section types. The names *VAL1* and *VAL2* are renamed for each type of section.

Notes

The **SECPLOT** command is valid only for BEAMS and SHELLS.

SECPLOT cannot display the plot of an ASEC (arbitrary section) subtype.

BEAMS

Plots the geometry of the beam section to scale depicting the centroid, shear center, and origin. **SECPLOT** also lists various section properties such as *Iyy*, *Iyz*, and *Izz*.

Data to be supplied in the value fields:

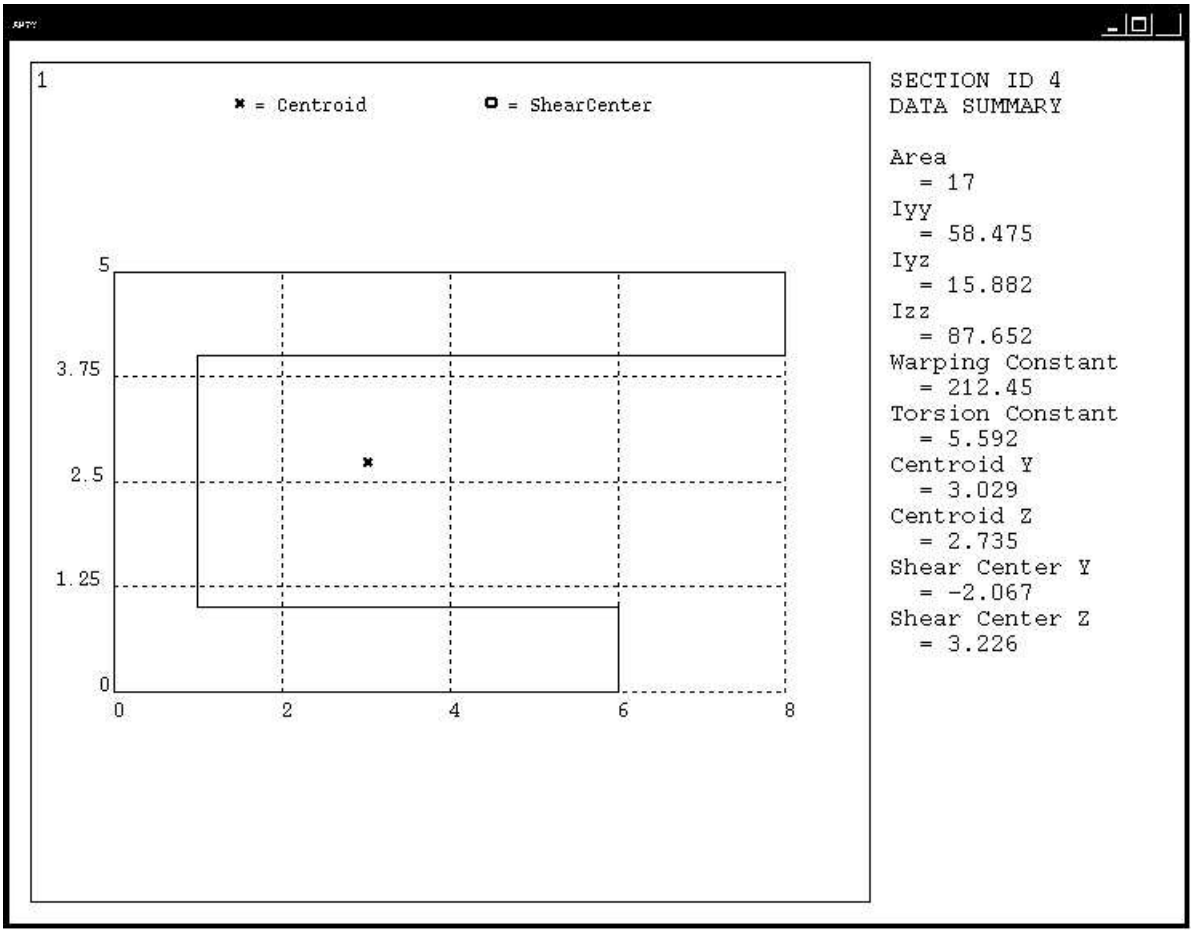
MESHKEY
Displays or suppresses display of beam section mesh.

0 --
Does not display beam section mesh.

1 --
Displays beam section mesh.

A sample section plot for the CHAN section subtype is shown below.

SECPLOT Beams Sample



SHELLS

Plots the layer arrangement of the shell section showing the layer material and orientation.

Data to be supplied in the value fields:

LAYR1, LAYR2
: Range of layer numbers to be displayed. If *LAYR1* is greater than *LAYR2*, a reversed order display is produced. Up to 20 layers may be displayed at a time. *LAYR1* defaults to 1. *LAYR2* defaults to *LAYR1* if *LAYR1* is input or to the number of layers (or to 19+*LAYR1*, if smaller) if *LAYR1* is not input.

A sample section plot for the SHELL section type is shown below.

SECPLOT Shells Sample

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Plot Section
Main Menu>Preprocessor>Sections>Shell>Plot Section

SECREAD, *Fname*, *Ext*, --, *Option*

Reads a customized beam section library or a user-defined beam section mesh into ANSYS.

PREP7: Cross Sections

MP ME ST <> <> PR <> <> <> PP ED

Fname

Section library file name and directory path containing the section library file (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

When the **SECREAD** command is given without a directory path, the command searches for a section library in the following order:

- The user's home directory
- The current working directory
- The path specified by the **/SECLIB** command

The file name defaults to **Jobname** if *Fname* is left blank.

Ext
Filename extension (8 character maximum).

The extension defaults to SECT if *Ext* is left blank.

--
Unused field

Option

LIBRARY
Reads in a library of sections and their associated section data values; the default. A section library may be created by editing the section-defining portions of the **Jobname.LOG** file and saving it with a **.SECT** suffix.

MESH
Reads in a user mesh section file containing the cell connectivity, cell flags, and nodal coordinates for the current beam section of subtype MESH as defined by **SECTYPE**. See the Notes section of this command description for details about user mesh section files. **SECWRITE** builds mesh files based on 2-D models you create.

Notes

Sample User Section Cell Mesh File

Here are excerpts from a sample user section mesh file for a section with 75 nodes, 13 cells, and 9 nodes per cell for a two-hole box section. Illustrations of the two-hole box section and the cell mesh for it appear later in this command description.

First Line:

```
75   13
```

Cells Section:

```
        1   3   25   23   2   14   24   12   13   1
        3   5   27   25   4   16   26   14   15   1
        5   7   29   27   6   18   28   16   17   1
        7   9   31   29   8   20   30   18   19   1
        9   11  33   31   10  22   32   20   21   1
        23  25  45   43   24  36   44   34   35   1
...
        49  51  73   71   50  62   72   60   61   1
        51  53  75   73   52  64   74   62   63   1
0       0.0          0.0
0       0.025        0.0
0       0.05         0.0
0       5.0175       0.0
```

Nodes Section:

```
...
0     19.98         10.00
0     20.00         10.00
```

The mesh file is divided into three sections: the First Line, the Cells Section, and the Nodes Section. Here are brief descriptions of the contents of each.

First Line: The First Line defines the number of nodes and the number of cells for the mesh.

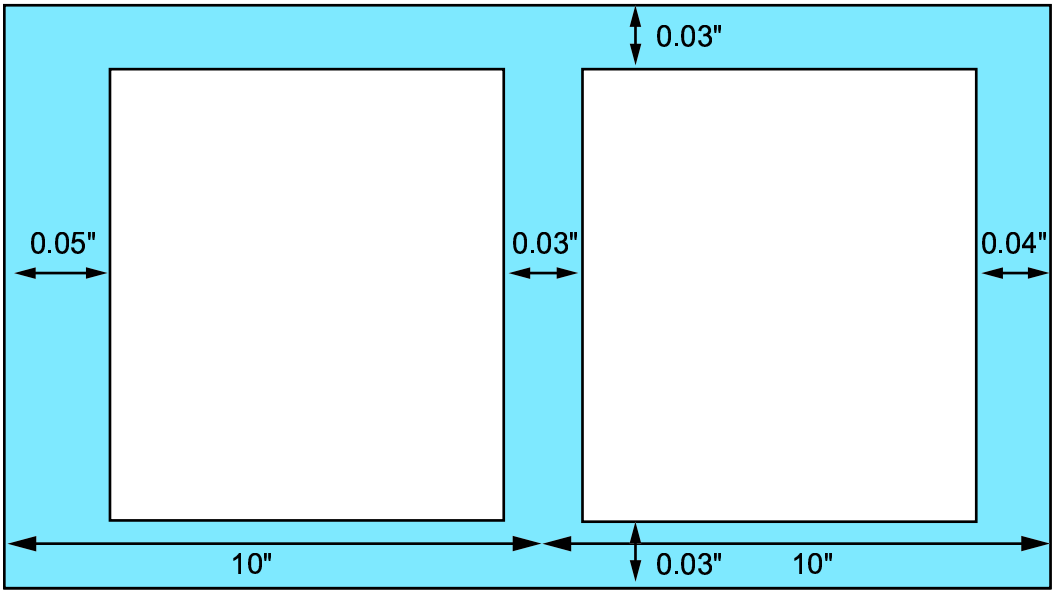
Cells Section: The Cells Section contains as many lines as there are cells. In this example, there are thirteen cells, so there are thirteen lines in this section. In each line, the number "1" that follows the cell connectivity information is the material number.

Cell nodal connectivity must be given in a counterclockwise direction, with the center node being the ninth node. For details, see Cell Mesh for the Two-hole Box Section.

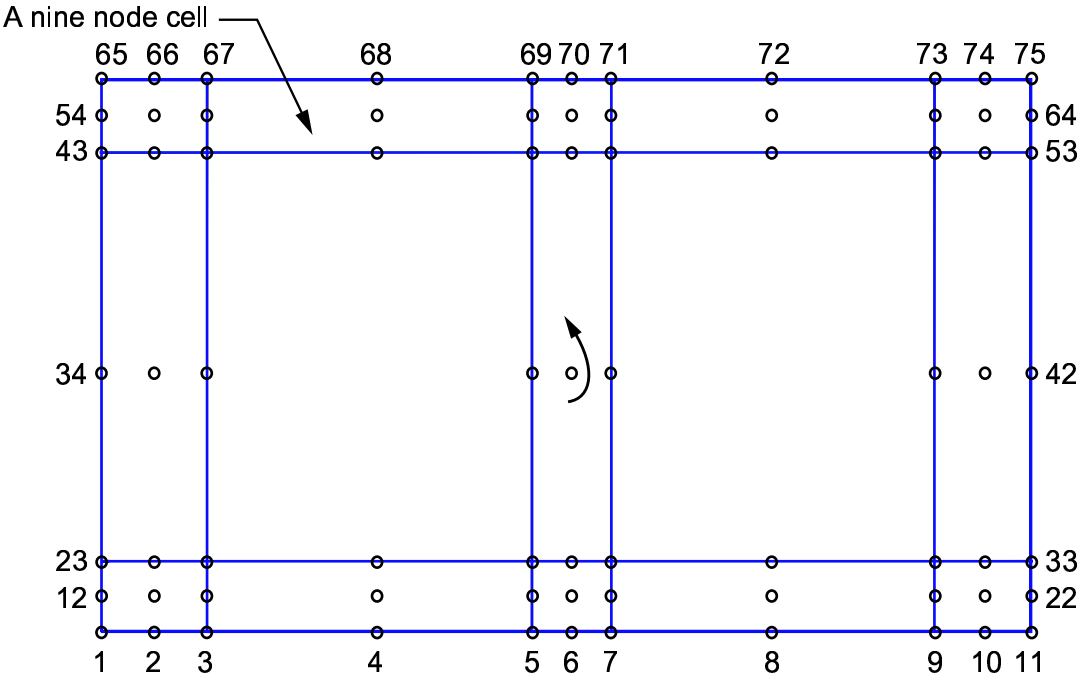
Nodes Section: The Nodes Section contains as many lines as there are nodes. In this example, there are 75 nodes, so there are a total of 75 lines in this section. Each node line contains the node's boundary flag, the Y coordinate of the node, and the Z coordinate of the node. Currently, all node boundary flags appear as 0s in a cell mesh file (as illustrated in Two-hole Box Section). Since all node boundary flags are 0, **SECREAD** ignores them when it reads a cell mesh file into ANSYS.

There cannot be any gaps in the node numbering of a cell mesh. The nodes in a cell mesh must be numbered consecutively, with the first node having a node number of 1, and the last node having a node number that is equal to the maximum number of nodes in the cell mesh.

Two-hole Box Section



Cell Mesh for the Two-hole Box Section



Menu Paths

Main Menu>Preprocessor>Sections>Beam>Custom Sections>Read Sect Mesh
Main Menu>Preprocessor>Sections>Section Library>Import Library

---

SECSTOP, *dof*, *MINVALUE*, *MAXVALUE*, *dof*, *MINVALUE*, *MAXVALUE*, *dof*, *MINVALUE*, *MAXVALUE*
Specifies stops on the components of relative motion in a joint element.

PREP7: Cross Sections

MP ME ST <> <> PR <> <> <> PP ED

dof
Local degree of freedom to be stopped.

MINVALUE
Low end of the range of allowed movement for the specified DOF.

MAXVALUE
High end of the range of allowed movement for the specified DOF.

Notes

Stops restrict motion in a DOF; motion beyond the MINVALUE or MAXVALUE is prevented (motion away from a limit is allowed). You can specify up to three stops. If necessary, you can repeat the command.

Menu Paths

Main Menu>Preprocessor>Sections>Joints>Add / Edit

SECTYPE, *SECID*, *Type*, *Subtype*, *Name*, *REFINEKEY*

Associates section type information with a section ID number.

PREP7: Cross Sections

MP ME ST <> <> PR <> <> <> PP ED

SECID

Section identification number.

Type

BEAM

Defines a beam section. See subtypes below.

TAPER

Defines a tapered beam section. The sections at the end points must be topologically identical.

SHELL

Defines a shell.

PRETENSION

Defines a pretension section.

JOINT

Defines a joint section.

Subtype

When *Type* = BEAM, the possible beam sections that can be defined for *Subtype* are:

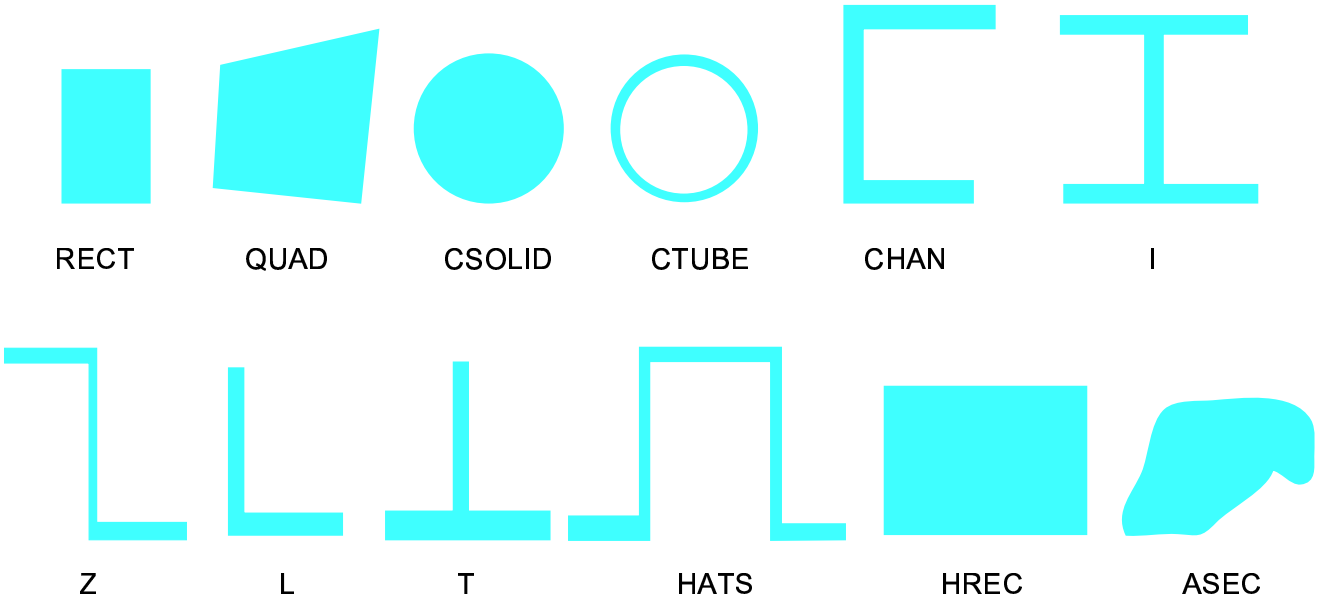
RECT	Rectangle
QUAD	Quadrilateral
CSOLID	Circular solid
CTUBE	Circular tube
CHAN	Channel
I	I-shaped section
Z	Z-shaped section
L	L-shaped section
T	T-shaped section
HATS	Hat-shaped section
HREC	Hollow rectangle or box
ASEC	Arbitrary section -- integrated cross-section inertia properties supplied by user
MESH	User-defined mesh -- see the **SECREAD** command for more information about this data

When *Type* = JOINT, the possible joint sections that can be defined for *Subtype* are:

UNIV	Universal joint
REVO	Revolute joint

The following figure shows the shape of each cross section subtype:

Cross Section Subtypes



See the **SECDATA** command for detailed illustrations of the BEAM section subtypes and their associated geometric data.

Name
An 8-character name for the section. *Name* can be a string such as "W36X210" or "HP13X73" for beam sections. *Name* must follow ANSYS naming conventions. Section names may contain letters and numbers, but cannot contain punctuation, special characters, or spaces.

REFINEKEY
Sets mesh refinement level for thin-walled beam sections. Values are 0 (the default - no mesh refinement) to 5 (high level of mesh refinement).

Notes

SECTYPE sets the section ID number, section type, and subtype for a section. If the section ID number is not specified, ANSYS increments the highest section ID number currently defined in the database by one. A previously-defined section with the same identification number will be redefined. The geometry data describing this section type is defined by a subsequent **SECDATA** command. Define the offsets by a subsequent **SECOFFSET** command. For a beam section, ANSYS builds a numeric model using a nine node cell for determining the properties (Ixx, Iyy, etc.) of the section and for the solution to the Poisson's equation for torsional behavior. The **SLIST** command lists the section properties, and the **SECPLOT** command displays the section to scale. The **SECNUM** command assigns the section ID number to a subsequently-defined beam element. See Beam Analysis and Cross Sections in the *ANSYS Structural Analysis Guide* for examples using the section commands.

When **SECTYPE** is used to define a tapered beam section (*Type* = TAPER), two subsequent **SECDATA** commands are required (one for each end section). Section ends must be topologically identical (same *Subtype*, number of cells and material IDs). See Defining a Tapered Beam in the *ANSYS Structural Analysis Guide* for more information.

The *PRETENSION* section options of **SECTYPE** and **SECDATA** are documented mainly to aid in the understanding of data written by **CDWRITE**. We advise that you generate pretension sections using **PSMESH**.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Common Sections
Main Menu>Preprocessor>Sections>Beam>Custom Sections>Read Sect Mesh
Main Menu>Preprocessor>Sections>Beam>Taper Sections>By Picked Nodes
Main Menu>Preprocessor>Sections>Beam>Taper Sections>By XYZ Location
Main Menu>Preprocessor>Sections>Joints>Add / Edit
Main Menu>Preprocessor>Sections>Shell>Add / Edit

SECWRITE, *Fname*, *Ext*, --, *ELEM_TYPE*

Creates an ASCII file containing user mesh section information.

PREP7: Cross Sections

MP ME ST <> <> PR <> <> <> PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname** if *Fname* is left blank.

Ext
Filename extension (8 character maximum).

The extension defaults to SECT if *Ext* is left blank.

--
Unused field

ELEM_TYPE
Element type attribute pointer (**ET**) for the elements that are part of the section. See **SECREAD** for a detailed description.

Notes

Before creating a user mesh file, you must create a model using ANSYS 2-D meshing capabilities. Use PLANE82 or MESH200 with KEYOPT(1) = 7 (quadrilateral with 8 nodes option) to model the cells. **SECWRITE** creates an ASCII file that contains information about the nodes and cells that describe a beam section. For detailed information on how to create a user mesh file, see Managing Cross Section and User Mesh Libraries in the *ANSYS Structural Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Sections>Custom Sectns>Write From Areas

---

SED, *SEDX*, *SEDY*, *SEDZ*

Defines the excitation direction for a single-point response spectrum.

SOLUTION: Spectrum Options

MP ME ST <> <> PR <> <> <> PP ED

SEDX, *SEDY*, *SEDZ*

Global Cartesian coordinates of a point that defines a line (through the origin) corresponding to the excitation direction. For example: 0.0, 1.0, 0.0 defines global Y as the spectrum direction. Spectrum values are not scaled with this input.

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>DDAM Options
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Settings
Main Menu>Solution>Load Step Opts>Spectrum>DDAM Options
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Settings

SEDLIST, *Sename*, *KOPT*

Lists the DOF solution of a superelement after the use pass.

PREP7: Superelements

MP ME ST <> <> <> <> <> <> PP ED

Sename

Name of the superelement on **File.DSUB** to be listed. If a number, it is the element number of the superelement as used in the use pass. If ALL, list results for all superelements.

KOPT

List key:

0
: List summary data only.

1
: List full contents. Be aware that the listing may be extensive.

Notes

Lists the degree of freedom solution of a superelement after the substructure use pass. Results may be listed for any superelement on **File.DSUB**.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>List Results>Superelem DOF
Utility Menu>List>Results>Superelem DOF Solu

SEEXP, *Sename*, *Usefil*, *Imagky*

Specifies options for the substructure expansion pass.

SOLUTION: Analysis Options

MP ME ST <> <> <> <> <> <> PP ED

Sename
Name of the superelement matrix file created by the substructure generation pass (***Sename*.SUB**). Defaults to the initial jobname **File**. If a number, it is the element number of the superelement as used in the use pass.

Usefil
Name of the file containing the superelement degree-of-freedom (DOF) solution created by the substructure use pass (***Usefil*.DSUB**).

Imagky
Key to specify use of the imaginary component of the DOF solution. Applicable only if the use pass is a harmonic (**ANTYPE**,HARMIC) analysis:

OFF
Use real component of DOF solution (default).

ON
Use imaginary component of DOF solution.

Notes

Specifies options for the expansion pass of the substructure analysis (**ANTYPE**,SUBSTR). If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in **/PREP7**.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Expand Superelem
Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Expand Superelem

---

/SEG, *Label*, *Aviname*, *DELAY*

Allows graphics data to be stored in the local terminal memory.

GRAPHICS: Set Up
DISPLAY: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Label
Storage key:

SINGL
Store subsequent display in a single segment (overwrites last storage).

MULTI
Store subsequent displays in unique segments [**ANIM**].

DELET
Delete all currently stored segments.

OFF
Stop storing display data in segments.

STAT
Display segment status.

PC
This option only applies to PC versions of ANSYS and only when animating via the AVI movie player (i.e., **/DEVICE**,ANIM,2). This command appends frames to the **File.AVI**, so that the animation goes in both directions (i.e., forward--backward--forward). You must have a current animation file to use this option.

Aviname
Name of the animation file that will be created when each frame is saved. The **.AVI** extension is applied automatically. Defaults to **Jobname.AVI** if no filename is specified.

DELAY
Delay factor between each frame, in seconds. Defaults to 0.015 seconds if no value is specified.

Command Default

No segment storage.

Notes

Allows graphics data to be stored in the terminal local memory (device-dependent). Storage occurs concurrently with the display.

Although the information from your graphics window is stored as an individual segment, you cannot plot directly (**GPLOT**) from the segment memory.

For the DISPLAY program, the *Aviname* and *DELAY* fields are ignored.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Redirect Plots>Delete Segments
Utility Menu>PlotCtrls>Redirect Plots>Segment Status
Utility Menu>PlotCtrls>Redirect Plots>To Segment Memory

SEGEN, *Mode*, *nSuper*, *mDof*, *stopStage*
Automatically generate superelements.

SOLUTION: Analysis Options

MP ME ST <> <> PR <> <> <> PP ED

Mode
Specify action to take (must be specified as one of the following):

AUTO
Turn on feature.

OFF
Turn off feature.

nSuper
Number of superelements to create. The minimum number of superelements is 2, and the maximum number of superelements is 999. Note that the number of requested superelements may not be the same as the number of defined superelements (see Notes for more details).

mDof
Specifies whether to use the master DOF defined by the user.

YES
Use master DOF defined by the user with the **M** command.

NO
Use the master DOF defined by the automatic generation process. Be aware that this option can generate a large number of master DOFs (see Notes for more details).

stopStage
Specifies when to stop the automatic superelement generation process.

PREVIEW
Preview the superelements only; stop after creating the domains which will become the superelements, and after creating master DOF on the interfaces between each domain.

GEN
Create (generate) the superelements.

Command Default

No command default. If the command is issued, a *Mode* must be specified. By default, if the AUTO mode is chosen, the **SEGEN** command automatically creates 10 superelements and defines master DOFs at all DOFs on the interfaces between superelements.

Notes

This command can be used to quickly generate a set of superelements. Each superelement is created in a separate file (**jobnameXXX.sub**, where *XXX* is a positive number from 1 to 999).

Due to the heuristics in the automatic domain decomposer, which is used to define the domains that will become superelements, the number of defined superelements may exceed the number of requested superelements. Use the *mDof* and *stopStage* options to determine exactly how many superelements will be created, the interfaces between each superelement, and where master DOF will be defined. With the **/PNUM**,DOMAIN command, you can graphically (visually) preview the elements in each superelement. Then, if required, you can add additional master DOF to (or remove from) the boundaries of the superelements. Use the **SEGEN** command again with *stopStage* = GEN to actually create the superelements.

ANSYS automatically defines master DOF at each of the following: all interface DOF between superelements, all DOF attached to contact elements (TARGE169 to CONTA175), and all DOF associated with nodes having a point load defined. Note that for regular superelements, all interface DOFs must be defined as master DOFs for the correct solution to be obtained. However, for CMS superelements, some of the interface DOFs can be removed without a significant loss of accuracy.

For the case when *mDof* = YES, you should select the preview option first (*stopStage* = PREVIEW) to verify exactly how many superelements will be created and where the superelement boundaries are located. If more superelements will be created than were requested, you should define master DOF on the interface(s) between all superelements.

This command is valid only for substructuring analyses (**ANTYPE**,SUBSTR). Either the frontal or sparse solver can be selected. Use **SEOPT** to specify any options for all of the superelements (e.g., which matrices to reduce), and possibly **CMSOPT** for any CMS substructuring analysis. Note that the created superelements will follow the current **/FILNAME** instead of *SENAME* from **SEOPT**. Also, only one load vector will be written to each **.SUB** file. Multiple load steps are not supported with the automatic superelement generation process.

During the actual creation of the superelements, the output is redirected to **jobname.autoTemp**.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

SELIST, *Sename*, *KOPT*

Lists the contents of a superelement matrix file.

PREP7: Superelements

MP ME ST <> <> <> <> <> <> PP ED

Sename
Name of the superelement matrix file created by the substructure generation pass (***Sename*.SUB**). Defaults to the current **Jobname**. If a number, it is the element number of the superelement as used in the use pass.

KOPT
List key:

0
List summary data only.

1
List contents, except load vectors and matrices.

2
List contents, except matrices.

3
List full contents. Be aware that the listing may be extensive.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Other>Superelem Data

SELM

Specifies "Superelements" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Superelements

SENERGY, *OPT*, *ANTYPE*

Determines the stored magnetic energy or co-energy.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

OPT

Item to be calculated:

0
: Stored magnetic energy.

1
: Stored magnetic co-energy.

ANTYPE

Analysis type:

0
: Static or transient.

1
: Harmonic.

Notes

SENERGY invokes an ANSYS macro which calculates the stored magnetic energy or co-energy for all selected elements. (For a harmonic analysis, the macro calculates a time-averaged (rms) stored energy.) A summary table listing the energy by material number is produced. The energy density is also calculated and stored on a per-element basis in the element table [**ETABLE**] with the label MG_ENG (energy density) or MG_COENG (co-energy density). The macro erases all other items in the element table [**ETABLE**] and only retains the energy density or co-energy density. Use the **PLETAB** and **PRETAB** commands to plot and list the energy density. The macro is valid for static and low-frequency magnetic field formulations. The macro will not calculate stored energy and co-energy for the following cases:

- Orthotropic nonlinear permanent magnets.
- Orthotropic nonlinear permeable materials.
- Temperature dependent materials.

SENERGY is restricted to MKSA units.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Element Based>Co-Energy
Main Menu>General Postproc>Elec&Mag Calc>Element Based>Energy

---

SEOPT, *Sename*, *SEMATR*, *SEPR*, *SESST*, *EXPMTH*

Specifies substructure analysis options.

SOLUTION: Analysis Options

MP ME ST <> <> <> <> <> <> PP ED

Sename
Name assigned to the superelement matrix file. The matrix file will be named ***Sename*.SUB**. This field defaults to *Fname* on the **/FILNAME** command.

SEMATR
Matrix generation key:

1
Generate stiffness (or conductivity) matrix (default).

2
Generate stiffness and mass (or conductivity and specific heat) matrices.

3
Generate stiffness, mass and damping matrices.

SEPR
Print key:

0
Do not print superelement matrices or load vectors.

1
Print both load vectors and superelement matrices.

2
Print load vectors but not matrices.

SESST
Stress stiffening key:

0
Do not save space for stress stiffening in a later run.

1
Save space for the stress stiffening matrix (calculated in a subsequent generation run after the expansion pass).

EXPMTH
Expansion method for expansion pass:

BACKSUB
Save necessary triangularized matrix files for backsubstitution during subsequent expansion passes (default). This normally results in a large usage of disk space

RESOLVE
Do not save triangularized matrix files. Global stiffness matrix will be reformed during expansion pass. This option provides an effective way to save disk space usage.

Notes

Specifies substructure analysis options (**ANTYPE**,SUBSTR). If used in SOLUTION, this command is valid only within the first load step.

The sparse solver cannot be used to generate unsymmetric superelement matrices or to generate superelement matrices with damping. If either case is encountered, the program will automatically switch to the frontal solver.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

SESYMM, *Sename*, *Ncomp*, *INC*, *File*, *Ext*, --

Performs a symmetry operation on a superelement within the use pass.

PREP7: Superelements

MP ME ST <> <> <> <> <> <> PP ED

Sename
Name of the superelement matrix file created by the substructure generation pass (***Sename*.SUB**). Defaults to the current **Jobname**. If a number, it is the element number of a previously defined superelement in the current use pass.

Ncomp
Symmetry key:

X
X symmetry (default).

Y
Y symmetry.

Z
Z symmetry.

INC
Increment all nodes in the superelement by *INC*.

File
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

This field must be input.

Ext
Filename extension (8 character maximum).

The extension defaults to SUB.

--
Unused field

Notes

Performs a symmetry operation on a superelement within the substructure use pass by reversing the sign of component *Ncomp* in the global Cartesian coordinate system. The node numbers are incremented by *INC*. The new superelement is written to **File.SUB** in the current directory (by default). All master node nodal coordinate systems must be global Cartesian (no rotated nodes allowed).

The maximum number of transformations for a given superelement is five (including **SETRAN**, **SESYMM**, and the large rotation transformation if **NLGEOM** is ON in the use pass).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>By Reflection

---

SET, *Lstep*, *SBSTEP*, *FACT*, *KIMG*, *TIME*, *ANGLE*, *NSET*, ORDER
Defines the data set to be read from the results file.

POST1: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Lstep
Load step number of the data set to be read (defaults to 1):

N
Read load step *N*.

FIRST
Read the first data set (*SBSTEP* and *TIME* are ignored).

LAST
Read the last data set (*SBSTEP* and *TIME* are ignored).

NEXT
Read the next data set (*SBSTEP* and *TIME* are ignored). If at the last data set, the first data set will be read as the next.

PREVIOUS
Read the previous data set (*SBSTEP* and *TIME* are ignored). If at the first data set, the last data set will be read as the previous.

NEAR
Read the data set nearest to *TIME* (*SBSTEP* is ignored). If *TIME* is blank, read the first data set.

LIST
Scan the results file and list a summary of each load step. (*FACT*, *KIMG*, *TIME* and *ANGLE* are ignored.)

SBSTEP
Substep number (within *Lstep*). For the Buckling (**ANTYPE**,BUCKLE) analysis or the Modal (**ANTYPE**,MODAL) analysis, the substep corresponds to the mode number. Defaults to last substep of load step (except for **ANTYPE**,BUCKLE or MODAL). If *Lstep* = LIST, *SBSTEP* = 0 or 1 lists the basic step information, whereas *SBSTEP* = 2 also lists the load step title, and labels imaginary data sets if they exist. Default maximum is 1000. When the number of substeps exceeds this limit, you need to issue **SET**,*Lstep*,LAST to bring in the 1000th load step. Use **/CONFIG** to increase the limit.

FACT
Scale factor applied to data read from the file. If zero (or blank), a value of 1.0 is used. A nonzero factor excludes non-summable items (see the *ANSYS Basic Analysis Guide*). Harmonic velocities or accelerations may be calculated from the displacement results from a Modal (**ANTYPE**,MODAL) or Harmonic Response (**ANTYPE**,HARMIC) analyses. If *FACT* = VELO, the harmonic velocities (v) are calculated from the displacements (d) at a particular frequency (f) according to the relationship $v = 2\pi fd$. Similarly, if *FACT* = ACEL, the harmonic accelerations (a) are calculated as $a = (2\pi f)^2 d$.

KIMG
Used only with results from complex analyses.

0
Store real part of complex solution.

1
Store imaginary part.

Note — For damped modal solutions, the imaginary part of the eigenvalue represents the frequency of the system.

TIME
Time-point identifying the data set to be read. For the harmonic response analyses, time corresponds to the frequency. For the buckling analysis, time corresponds to the load factor. Used only in the following cases: If *Lstep* = NEAR, read the data set nearest to *TIME*. If both *Lstep* and *SBSTEP* are zero (or blank), read data set at time = *TIME*. Do not use *TIME* to identify the data set to be read if you used the arc-length method [**ARCLEN**] in your solution. If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets. Solution items not written to the results file [**OUTRES**] for either data set will result in a null item after data set interpolation. If *TIME* is beyond the last time point on the file, the last time point will be used.

ANGLE
Circumferential location (0.0 to 360°). Defines the circumferential location for the harmonic calculations used when reading from the results file. The harmonic factor (based on the circumferential angle) is applied to the harmonic elements (PLANE25, PLANE75, PLANE78, FLUID81, PLANE83, and SHELL61) of the load case. See the *ANSYS, Inc. Theory Reference* for details.

Note — The factored values of applied constraints and loads will overwrite any values existing in the database. If *ANGLE* = NONE, all harmonic factors are set to 1 and postprocessing will yield the solution output. When using *ANGLE* = NONE with MODE > 0, the combined stresses and strains are not valid. The default value of *ANGLE* is 0.0, but if the **SET** command is not used, the effective default is NONE.

NSET
Data set number of the data set to be read. If a positive value for *NSET* is entered, *Lstep*, *SBSTEP*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by **SET**,LIST.

ORDER
Sorts the harmonic index results in ascending order of eigenfrequencies or buckling load multipliers. This value applies to cyclic symmetry buckling and modal analyses only, and is valid only when *Lstep* = FIRST, LAST, NEXT, PREVIOUS, NEAR or LIST.

Notes

Defines the data set to be read from the results file into the database. Various operations may also be performed during the read operation. The database must have the model geometry available (or use the **RESUME** command before the **SET** command to restore the geometry from **File.DB**). Values for applied constraints [**D**] and loads [**F**] in the database will be replaced by their corresponding values on the results file, if available. (See the description of the **OUTRES** command.) In a single load step analysis, these values are usually the same, except for results from harmonic elements. (See the description of the *ANGLE* value above.)

In an interactive run, the sorted list (ORDER option) is also available for results-set reading via a GUI pick option.

Menu Paths

Main Menu>General Postproc>List Results>Detailed Summary
Main Menu>General Postproc>List Results>Detailed Summary (Freq ordered)
Main Menu>General Postproc>Read Results>By Load Step
Main Menu>General Postproc>Read Results>By Pick
Main Menu>General Postproc>Read Results>By Pick (Freq ordered)
Main Menu>General Postproc>Read Results>First Set
Main Menu>General Postproc>Read Results>Last Set
Main Menu>General Postproc>Read Results>Next Set
Main Menu>General Postproc>Read Results>Previous Set
Main Menu>General Postproc>Results Summary
Main Menu>General Postproc>Results Summary (Freq ordered)
Utility Menu>List>Results>Load Step Summary

SETFGAP, *GAP*, *ROPT*, *--*, *PAMB*, *ACF1*, *ACF2*, *PREF*, *MFP*

Updates or defines the real constant table for squeeze film elements.

PREP7: Real Constants

MP ME <> <> <> <> <> <> <> PP ED

GAP
Gap separation.

ROPT
Real constant set option.

0
Creates separate real constant sets for each selected element with the specified real constant values (default).

1
Updates existing real constant sets. The gap separation is updated from displacement results in the database. Other real constants are updated as specified in the command input parameters.

--
Unused field

PAMB
Ambient pressure.

ACF1, *ACF2*
Accommodation factor 1 and 2.

PREF
Reference pressure for mean free path.

MFP
Mean free path.

Notes

This command is used for large signal cases to update the gap separation real constant on a per-element basis. Issue this command prior to solution using the default *ROPT* value to initialize real constant sets for every fluid element. After a solution, you can re-issue the command to update the real constant set for a subsequent analysis. See Chapter 16, " Thin Film Analysis" for more information on thin film analyses.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Real Constants>ThinFilm
Main Menu>Preprocessor>Real Constants>ThinFilm
Main Menu>Solution>Load Step Opts>Other>Real Constants>ThinFilm

---

SETRAN, *Sename*, *KCNTO*, *INC*, *File*, *Ext*, *--*, *DX*, *DY*, *DZ*, *NOROT*

Creates a superelement from an existing superelement.

PREP7: Superelements

MP ME ST <> <> <> <> <> <> PP ED

Sename
Name of the file containing the original superelement matrix created by the generation pass (***Sename*.SUB**). Defaults to the current **Jobname**. If *Sename* is a number, it is the element number of a previously defined superelement in the current use pass.

KCNTO
Reference number of coordinate system where the superelement is to be transferred to. Defaults to the global Cartesian system. Transfer occurs from the active coordinate system.

INC
Node offset. Defaults to zero. All new element node numbers will be offset from those on the original by *INC*.

File
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

This field must be input.

Ext
Filename extension (8 character maximum).

The extension defaults to SUB.

--
Unused field

DX, *DY*, *DZ*
Node location increments in the global Cartesian coordinate system. Defaults to zero.

NOROT
Node rotation key:

0
The nodal coordinate systems of the transferred superelement will also be rotated into the *KCNTO* system (i.e., the nodal coordinate systems will rotate with the superelement). The superelement matrices are not modified.

1
The nodal coordinate systems will not be rotated (they will remain fixed in their original global orientation). The superelement matrices and load vectors are modified if any rotations are done.

Note — If this option is chosen for models with displacement degrees of freedom, *and KCNTO is not the active system*, the superelement *Sename* must have six MDOF at each node.

Notes

Creates a superelement from an existing superelement and writes the new element to a file. The new element may then be read in (during the use pass) with an **SE** command. Superelements may be created from the original by transferring its (the original's) geometry from the active coordinate system into another coordinate system (*KCNTO*), by offsetting its geometry in the global Cartesian coordinate system (*DX*, *DY*, and *DZ*), or by offsetting its node numbers (*INC*). All three methods may be used in combination. If both the geometry transfer and the geometry offset are used, the transfer is done first.

The maximum number of transformations for a given superelement is five (including **SETRAN**, **SESYMM**, and the large rotation transformation if **NLGEOM** is ON in the use pass).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>By CS Transfer
Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>By Geom Offset

---

SEXP, *LabR*, *Lab1*, *Lab2*, *EXP1*, *EXP2*

Forms an element table item by exponentiating and multiplying.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

LabR
Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

Lab1
First labeled result item in operation.

Lab2
Second labeled result item in operation (may be blank).

EXP1
Exponent applied to *Lab1*.

EXP2
Exponent applied to *EXP2*.

Notes

Forms a labeled result item (see **ETABLE** command) for the selected elements by exponentiating and multiplying two existing labeled result items according to the operation:

$$LabR = (|Lab1|^{EXP1}) \times (|Lab2|^{EXP2})$$

Roots, reciprocals, and divides may also be done with this command.

Menu Paths

Main Menu>General Postproc>Element Table>Exponentiate

SF, *Nlist*, *Lab*, *VALUE*, *VALUE2*
Specifies surface loads on nodes.

SOLUTION: FE Surface Loads

MP ME ST <> <> PR EM <> FL PP ED

Nlist
Nodes defining the surface upon which the load is to be applied. Use the label ALL or P, or a component name. If ALL, all selected nodes [**NSEL**] are used (default). If P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Lab
Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *ANSYS Elements Reference*. Structural labels: PRES (pressure). Thermal labels: CONV (convection); HFLUX (heat flux); RAD (radiation); RDSF (surface-to-surface radiation). Acoustic Fluid labels: FSI (fluid-structure interaction flag); IMPD (impedance); PTOT (constant total pressure. Magnetic labels: MXWF (Maxwell force flag); MCI (magnetic circuit interface). Electric labels: CHRGS (surface charge density); MXWF (Maxwell force flag). Infinite element labels: INF (Exterior surface flag for INFIN110 and INFIN111). High-frequency electromagnetic labels: PORT (number 1 through 50 for a waveguide exterior port); SHLD (surface shielding properties); MXWF (Maxwell surface flag of equivalent source surface). Field-Surface Interface label: FSIN (field-surface interface number).

Note — Thermal labels CONV and HFLUX are mutually exclusive. For a fluid-solid interaction analysis, apply the field-surface interface flag (Label = FSIN) twice: once for the fluid side (FLUID141 or FLUID142 elements) and once for the solid side. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the FLUID129 or FLUID130 elements. For more information about these analyses and labels, see the *ANSYS Coupled-Field Analysis Guide* and the *ANSYS Elements Reference*.

VALUE
Surface load value or table name reference for specifying tabular boundary conditions. If *Lab* = PRES, *VALUE* is the real component of the pressure. If *Lab* = CONV, *VALUE* is typically the film coefficient and *VALUE2* (below) is typically the bulk temperature. If *Lab* = CONV and *VALUE* = -*N*, the film coefficient may be a function of temperature and is determined from the HF property table for material *N* [**MP**]. The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user-defined for some elements. To specify a table, enclose the table name in percent signs (%) (e.g., **SF**,NLIST,Lab,%*tabname*%). Use the ***DIM** command to define a table. If *Lab* = MCI, *VALUE* indicates current direction (-1; current flow into the element face (IN), +1; current flow out of the element face (OUT). If *Lab* = RAD, *VALUE* is surface emissivity. If *Lab* = PORT, *VALUE* is a port number representing a waveguide exterior port. The port number must be an integer between 1 and 50. If *Lab* = SHLD, *VALUE* is surface conductivity. If *Lab* = RDSF, *VALUE* is the emissivity value; the following conditions apply: If *VALUE* is between 0 and 1, apply a single value to the surface. If *VALUE*= -*N*, the emissivity may be a function of the temperature, and is determined from the EMISS property table for material *N* (**MP**). The material *N* does not need to correlate with the underlying solid thermal elements. If *Lab* = FSIN, *VALUE* is the interface number.

VALUE2
Second surface load value (if any). If *Lab* = PRES, *VALUE2* is the imaginary component of the pressure. Imaginary pressures can only be used by SURF153 and SURF154 and can only be used for a full harmonic response analysis (**HROPT**,FULL), or by a mode superposition harmonic response analysis (**HROPT**,MSUP) if the mode extraction method is Block Lanczos (**MODOPT**,LANB).

If *Lab* = CONV, *VALUE2* is typically the bulk temperature. To specify a table (*Lab* = CONV), enclose the table name in percent signs (%) (e.g., **SF**,*Nlist*,*Lab*,*VALUE*,%*tabname*%). Use the ***DIM** command to define a table.

If Lab = RAD, *VALUE2* is the ambient temperature. *VALUE2* is not used for PORT or FSIN.

If *Lab* = SHLD, *VALUE2* is relative permeability and defaults to 1.0.

If *Lab* = RDSF, *VALUE2* is the enclosure number. Radiation will occur between surfaces flagged with the same enclosure numbers. If the enclosure is open, radiation will also occur to ambient. If *VALUE2* is negative radiation direction is reversed and will occur inside the element for the flagged radiation surfaces. Negative value of enclosure number is applicable for FLUID141 and FLUID142 elements, to model radiation occurring between surfaces inside the fluid domain.

Notes

Individual nodes may not be entered for this command. The node list is to identify a surface and the *Nlist* field must contain a sufficient number of nodes to define an element surface. The loads are internally stored on element faces defined by the specified nodes. All nodes on an element face must be specified for the face to be used, and the element must be selected. If all nodes defining a face are shared by an adjacent face of another selected element, the face is not free and will not have a load applied. If more than one element can share the same nodes (for example, a surface element attached to a solid element), select the desired element type before issuing the **SF** command. The **SF** command applies only to area and volume elements. For shell elements, if the specified nodes include face one (which is usually the bottom face) along with other faces (such as edges), only face one is used. Where faces cannot be uniquely determined from the nodes, or where the face does not fully describe the load application, use the **SFE** command. A load key of 1 (which is typically the first loading condition on the first face) is used if the face determination is not unique. A uniform load value is applied over the element face.

See the **SFBEAM** command for applying surface loads to beam elements. See the **SFGRAD** command for an alternate tapered load capability. See the **SFFUN** command for applying loads from a node vs. value function. Also

see the **SFE** command for applying tapered loads on individual element faces. Use the **SFDELE** command to delete loads applied with this command. Use the **SFCUM** command to accumulate (add) surface loads applied with **SF**.

Tabular boundary conditions (*VALUE* = %*tabname*%) available only for the following surface load labels (*Lab* = PRES, CONV (film coefficient and/or bulk temperature) or HFLUX), boundary value and wetting status (VFRC), and surface emissivity and ambient temperature (RAD).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppImped_E>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppShield>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppInfinite>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppMaxwell>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppMCI>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Node Components
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Surface Rad>On Nodes
Main Menu>Preprocessor>Trefftz Domain>AppInfinite>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppImped_E>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppShield>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppInfinite>On Nodes
Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppMaxwell>On Nodes
Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Nodes
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Nodes
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppMCI>On Nodes
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Nodes
Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Node Components
Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Nodes
Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Nodes
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Nodes
Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Nodes
Main Menu>Solution>Define Loads>Apply>Thermal>Surface Rad>On Nodes

SFA, *AREA*, *LKEY*, *Lab*, *VALUE*, *VALUE2*

Specifies surface loads on the selected areas.

SOLUTION: Solid Surface Loads

MP ME ST <> <> PR EM <> <> PP ED

AREA

Area to which surface load applies. If ALL, apply load to all selected areas [**ASEL**]. If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component may be substituted for *AREA*.

LKEY

Load key associated with surface load (defaults to 1). Load keys (1,2,3, etc.) are listed under "Surface Loads" in the input data table for each element type in the *ANSYS Elements Reference*. *LKEY* is ignored if the area is the face of a volume region meshed with volume elements.

Lab

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each area type in the *ANSYS Elements Reference*. Structural label: PRES (pressure). Thermal labels: CONV (convection); HFLUX (heat flux); RAD (radiation); RDSF (surface-to-surface radiation). Fluid labels: FSI (fluid-structure interaction flag); IMPD (impedance). Magnetic label: MXWF (Maxwell force flag); MCI (magnetic circuit interface). Electric labels: MXWF (Maxwell force flag), CHRGS (surface charge density). Infinite element label: INF (Exterior surface flag for INFIN110 and INFIN111). High-frequency electromagnetic labels: PORT (number 1 through *50* for a waveguide exterior port); SHLD (surface shielding properties). Field-Surface interface label: FSIN (field-surface interface number).

Note — Thermal labels CONV and HFLUX are mutually exclusive. For a fluid-solid interaction analysis, apply the field-surface interface flag (Label = FSIN) twice: once for the fluid side (FLUID141 or FLUID142 elements) and once for the solid side. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the FLUID129 or FLUID130 elements. For more information about these analyses and labels, see the *ANSYS Coupled-Field Analysis Guide* and the *ANSYS Elements Reference*.

VALUE

Surface load value or table name reference for specifying tabular boundary conditions. If *Lab* = CONV, *VALUE* is typically the film coefficient and *VALUE2* (below) is typically the bulk temperature. If *Lab* = CONV and *VALUE* = *-N*, the film coefficient may be a function of temperature and is determined from the HF property table for material *N* [**MP**]. The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user-defined for some elements. To specify a table, enclose the table name in percent signs (%) (e.g., **SF**,NLIST,Lab,%*tabname*%). Use the ***DIM** command to define a table. If *Lab* = MCI, VALUE indicates current direction (-1; current flow into the element face (IN), +1; current flow out of the element face (OUT). If *Lab* = RAD, *VALUE* is the surface emissivity. If *Lab* = PORT, *VALUE* is a port number representing a waveguide exterior port. The port number must be an integer between 1 and 50. If *Lab* = SHLD, *VALUE* is surface conductivity. If *Lab* = RDSF, *VALUE* is the emissivity value; the following conditions apply: If *VALUE* is between 0 and 1, apply a single value to the surface. If *VALUE*= *-N*, the emissivity may be a function of the temperature, and is determined from the EMISS property table for material *N* (**MP**). The material *N* does not need to correlate with the underlying solid thermal elements. If *Lab* = FSIN, *VALUE* is the interface number and *LKEY* is ignored.

VALUE2

Second surface load value (if any). If *Lab* = CONV, *VALUE2* is typically the bulk temperature. If *Lab* = RAD*VALUE2* is ambient temperature.*VALUE2* is not used for other surface load labels. *VALUE2* is not used for PORT. If *Lab* = SHLD, *VALUE2* is relative permeability (defaults to 1.0). If *Lab* = RDSF, *VALUE2* is the enclosure number. Radiation will occur between surfaces flagged with the same enclosure numbers. If the enclosure is open,

radiation will also occur to ambient. If *VALUE2* is negative radiation direction is reversed and will occur inside the element for the flagged radiation surfaces. Negative value of enclosure number is applicable for FLUID141 and FLUID142 elements, to model radiation occurring between surfaces inside the fluid domain. To specify a table (*Lab* = CONV), enclose the table name in percent signs (%) (e.g., **SFA**,NLIST,Lab,VALUE,%*tabname*%). Use the ***DIM** command to define a table.

Notes

Surface loads may be transferred from areas to elements with the **SFTRAN** or **SBCTRAN** commands. See the **SFGRAD** command for an alternate tapered load capability.

Tabular boundary conditions (*VALUE* = %*tabname*%) available only for surface load labels (*Lab* = PRES, CONV (film coefficient and/or bulk temperature) or HFLUX), boundary value and wetting status (VFRC), and surface emissivity and ambient temperature (RAD).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppImped_E>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppShield>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppInfinite>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppMaxwell>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppMCI>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Surface Rad>On Areas
Main Menu>Preprocessor>Trefftz Domain>AppInfinite>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppImped_E>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppShield>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>EMPorts>Exterior Port>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppInfinite>On Areas
Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppMaxwell>On Areas
Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Areas
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Areas
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppMCI>On Areas
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Areas
Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Areas

Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Areas
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Areas
Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Areas
Main Menu>Solution>Define Loads>Apply>Thermal>Surface Rad>On Areas

---

SFACT, *TYPE*

Allows safety factor or margin of safety calculations to be made.

POST1: Element Table

MP ME ST DY <> PR <> <> <> PP ED

TYPE
Type of calculation:

0
No nodal safety factor or margin of safety calculations.

1
Calculate and store safety factors in place of nodal stresses.

2
Calculate and store margins of safety in place of nodal stresses.

Command Default

No nodal safety factor or margin of safety calculations.

Notes

Allows safety factor (SF) or margin of safety (MS) calculations to be made for the average nodal stresses according to:

SF = SALLOW/|Stress|

MS = (SALLOW/|Stress|) -- 1.0

Calculations are done during the display, select, or sort operation (in the active coordinate system [**RSYS**]) with results stored in place of the nodal stresses. Use the **PRNSOL** or **PLNSOL** command to display the results.

Note — The results are meaningful only for the stress (SIG1, SIGE, etc.) upon which **SALLOW** is based. Nodal temperatures used are those automatically stored for the node. Related commands are **SFCALC**, **SALLOW**, **TALLOW**.

Menu Paths

Main Menu>General Postproc>Safety Factor>Restore NodeStrs
Main Menu>General Postproc>Safety Factor>SF for Node Strs

SFADELE, *AREA*, *LKEY*, *Lab*

Deletes surface loads from areas.

SOLUTION: Solid Surface Loads

MP ME ST <> <> PR EM <> <> PP ED

AREA
Area to which surface load deletion applies. If ALL, delete load from all selected areas [**ASEL**]. If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *AREA*.

LKEY
Load key associated with surface load (defaults to 1). See the **SFA** command for details.

Lab
Valid surface load label. If ALL, use all appropriate labels. See the **SFA** command for labels.

Notes

Deletes surface loads (and all corresponding finite element loads) from selected areas.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Surface Ld>On All Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppImped_E>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppShield>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelExtPort>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppMCI>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Surface Rad>On Areas
Main Menu>Solution>Define Loads>Delete>All Load Data>All Surface Ld>On All Areas
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppImped_E>On Areas
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppShield>On Areas
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Areas
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelExtPort>On Areas
Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Areas
Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Areas
Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Areas

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Areas
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Areas
Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Areas
Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppMCI>On Areas
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Areas
Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Areas
Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Areas
Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Areas
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Areas
Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Areas
Main Menu>Solution>Define Loads>Delete>Thermal>Surface Rad>On Areas

---

SFALIST, *AREA*, *Lab*

Lists the surface loads for the specified area.

SOLUTION: Solid Surface Loads

MP ME ST <> <> PR EM <> <> PP ED

AREA

Area at which surface load is to be listed. If ALL (or blank), list for all selected areas [**ASEL**]. If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *AREA*.

Lab

Valid surface load label. If ALL (or blank), use all appropriate labels. See the **SFA** command for labels.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Surface Loads>On All Areas
Utility Menu>List>Loads>Surface Loads>On Picked Areas

SFBEAM, *ELEM*, *LKEY*, *Lab*, *VALI*, *VALJ*, *VAL2I*, *VAL2J*, *IOFFST*, *JOFFST*

Specifies surface loads on beam elements.

SOLUTION: FE Surface Loads

MP ME ST <> <> PR <> <> <> PP ED

ELEM

Element to which surface load is applied. If ALL, apply load to all selected beam elements [**ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted in *ELEM*.

LKEY

Load key associated with surface load (defaults to 1). Load keys (1, 2, 3, etc.) are listed under "Surface Loads" in the input table for each element type in the *ANSYS Elements Reference*. For beam elements, the load key defines the load orientation.

Lab
Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *ANSYS Elements Reference*. Structural labels: PRES (pressure).

VALI, *VALJ*
Surface load values at nodes I and J. If *VALJ* is blank, it defaults to *VALI*. If *VALJ* is zero, a zero is used.

VAL2I, *VAL2J*
Second surface load values at nodes I and J. Currently not used.

I0FFST
Offset distance from node I (toward node J) where *VALI* is applied.

J0FFST
Offset distance from node J (toward node I) where *VALJ* is applied. Offsets are available only for lateral surfaces of line elements having a KEYOPT(10) which is set. If no offsets are specified, the load is applied over the full element length. Values may also be input as length fractions, depending on the KEYOPT(10) setting. For example, for a line length of 5.0, an *I0FFST* distance of 2.0 or an *I0FFST* fraction of 0.4 represent the same point. If *J0FFST* = -1, *VALI* is assumed to be a point load at the location specified with *I0FFST* and *VALJ* is ignored.

Notes

Specifies surface loads on the selected beam elements. Use the **SFELIST** and **SFEDELE** commands to list and delete surface loads applied with this command.

Use the **SFCUM** command to accumulate (add) surface loads applied with **SFBEAM**. When **SFBEAM** follows **SFCUM**,ADD, the same *I0FFST* and *J0FFST* values must be used as on the previous **SFBEAM** command (for a given element face). Otherwise, the loads will not be accumulated. Leaving *I0FFST* and *J0FFST* blank will cause the previous offset values to be used (only when **SFBEAM** follows **SFCUM**).

For elements PIPE16, PIPE17, PIPE18, PIPE20, and PIPE60, tapered pressures are not recognized. Only constant pressures are supported for these elements.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Beams
Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Beams

---

SFCALC, *LabR*, *LabS*, *LabT*, *TYPE*

Calculates the safety factor or margin of safety.

POST1: Element Table

MP ME ST DY <> PR <> <> <> PP ED

LabR
Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

LabS
Labeled result item corresponding to the element stress.

LabT
Labeled result item corresponding to the element temperature.

TYPE
Type of calculation:

0 or 1
Use safety factor (SF) calculation.

2
Use margin of safety (MS) calculation.

3
Use 1/SF calculation.

Notes

Calculates safety factor (SF) or margin of safety (MS) as described for the **SFACT** command for any labeled result item (see **ETABLE** command) for the selected elements. Use the **PRETAB** or **PLETAB** command to display results. Allowable element stress is determined from the SALLOW-TALLOW table [**SALLOW**, **TALLOW**].

Menu Paths

Main Menu>General Postproc>Safety Factor>SF for ElemTable

SFCUM, *Lab*, *Oper*, *FACT*, *FACT2*

Specifies that surface loads are to be accumulated.

SOLUTION: FE Surface Loads

MP ME ST <> <> PR EM <> FL PP ED

Lab
Valid surface load label. If ALL, use all appropriate labels. Structural label: PRES (pressure). Thermal labels: CONV (convection); HFLUX (heat flux). Substructure label: SELV (Load vector number). Electric labels: CHRGS (surface charge density), MXWF (Maxwell force flag). Magnetic label: MXWF (Maxwell force flag). Infinite element: INF (Exterior surface flag for INFIN110 and INFIN111). Thermal labels CONV and HFLUX are mutually exclusive.

Oper
Accumulation key:

REPL
Subsequent values replace the previous values (default).

ADD
Subsequent values are added to the previous values.

IGNO
Subsequent values are ignored.

FACT
Scale factor for the first surface load value. A (blank) or '0' entry defaults to 1.0.

FACT2
Scale factor for the second surface load value. A (blank) or '0' entry defaults to 1.0.

Command Default

Replace previous values.

Notes

Allows repeated surface loads (pressure, convection, etc.) to be replaced, added, or ignored. Surface loads are applied with the **SF**, **SFE**, and **SFBEAM** commands. Issue the **SFELIST** command to list the surface loads. The operations occur when the next surface load specifications are defined. For example, issuing the **SF** command with a pressure value of 25 after a previous **SF** command with a pressure value of 20 causes the current value of that pressure to be 45 with the add operation, 25 with the replace operation, or 20 with the ignore operation. All new pressures applied with **SF** after the ignore operation will be ignored, even if no current pressure exists on that surface.

Scale factors are also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a pressure of 70. Scale factors are applied even if no previous values exist. Issue **SFCUM**,STAT to show the current label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

Note — The FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

SFCUM does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Surface Loads
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Surface Loads

---

SFDELE, *Nlist*, *Lab*

Deletes surface loads.

SOLUTION: FE Surface Loads

MP ME ST <> <> PR EM <> FL PP ED

Nlist
Label defining where to find the list of nodes:

ALL
Use all selected nodes [**NSEL**]. If P, use graphical picking in GUI. A component label may be substituted for *Nlist*.

Lab
Valid surface load label. If ALL, use all appropriate labels. See the **SF** command for labels.

Notes

Deletes surface loads as applied with the **SF** command. Loads are deleted only for the specified nodes on external faces of selected area and volume elements. For shell elements, if the specified nodes include face one (which is usually the bottom face) along with other faces (such as edges), only the loads on face one will be deleted. The element faces are determined from the list of selected nodes as described for the **SF** command. See the **SFEDELE** command for deleting loads explicitly by element faces.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppImped_E>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppShield>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelExtPort>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppMCI>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Node Components
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Surface Rad>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppImped_E>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppShield>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelExtPort>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Nodes
Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Nodes
Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Nodes
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Nodes
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Nodes
Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Nodes
Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppMCI>On Nodes
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Nodes
Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Node Components
Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Nodes
Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Nodes
Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Nodes
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Nodes
Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Nodes
Main Menu>Solution>Define Loads>Delete>Thermal>Surface Rad>On Nodes

SFE, *ELEM*, *LKEY*, *Lab*, *KVAL*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

Specifies surface loads on elements.

SOLUTION: FE Surface Loads

MP ME ST <> <> PR EM <> FL PP ED

ELEM

Element to which surface load applies. If ALL, apply load to all selected elements [**ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

LKEY

Load key associated with surface load (defaults to 1). Load keys (1,2,3, etc.) are listed under "Surface Loads" in the input data table for each element type in the *ANSYS Elements Reference*.

Lab

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *ANSYS Elements Reference*. Structural labels: PRES (pressure). Thermal labels: CONV (convection); HFLUX (heat flux); RAD (radiation); RDSF (surface-to-surface radiation). Fluid labels: FSI (fluid-structure interaction flag); VFRC (volume fraction for VOF method); IMPD (impedance). Magnetic labels: MXWF (Maxwell force flag). Infinite element labels: INF (Exterior surface flag for INFIN110 and INFIN111). Substructure label: SELV (load vector number). Electric labels: CHRGS (Surface Charge density); MXWF (Maxwell force flag). High-frequency electromagnetic labels: PORT (waveguide port number); SHLD (surface shielding properties). Field-Surface interface label: FSIN (field-surface interface number).

Note — Thermal labels CONV and HFLUX are mutually exclusive. For a fluid-solid interaction analysis, apply the field-surface interface flag (Label = FSIN) twice: once for the fluid side (FLUID141 or FLUID142 elements) and once for the solid side. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the FLUID129 or FLUID130 elements. For more information about these analyses and labels, see the *ANSYS Coupled-Field Analysis Guide* and the *ANSYS Elements Reference*.

KVAL

Value key. If *Lab* = PRES:

0 or 1
: *VAL1* through *VAL4* are used as real components of pressures.

2
: *VAL1* through *VAL4* are used as imaginary components of pressures.

Value key. If *Lab* = CONV:

0 or 1
: *VAL1* through *VAL4* are used as the film coefficients.

2
: *VAL1* through *VAL4* are the bulk temperatures.

Value key. If *Lab* = SHLD:

0 or 1
: *VAL1* through *VAL4* are used as the electrical conductivities.

2
: *VAL1* through *VAL4* are used as the relative permeabilities.

Value key. If *Lab* = RAD:

0 or 1
VAL1 through *VAL4* are used as the emissivities.

2
VAL1 through *VAL4* are ambient temperatures.

Value key. If *Lab* = RDSF:

0 or 1
VAL1 is the emissivity value between 0 and 1.

2
VAL1 is the enclosure number.

Value key. If *Lab* = VFRC:

0 or 1
VAL1 is the boundary load value. *VAL1* defaults to 1.

2
VAL1 is the wetting status. *VAL1* defaults to 0 or non-wetting.

If only one set of data is supplied (either emissivities or temperatures when *Lab* = RAD; or either film coefficients or temperatures when *Lab* = CONV; or either conductivity or relative permeability when *Lab* = SHLD), the other set of data defaults to previously specified values (or zero if not previously specified).

VAL1
First surface load value (typically at the first node of the face) or the name of a table for specifying tabular boundary conditions. Face nodes are listed in the order given for "Surface Loads" in the input data table for each element type in the *ANSYS Elements Reference*. For example, for SOLID45, the item 1-JILK associates *LKEY* = 1 (face 1) with nodes J, I, L, and K. Surface load value *VAL1* then applies to node J of face 1. To specify a table, enclose the table name in percent signs (%), e.g., %*tabname*%. Use the ***DIM** command to define a table. *VAL2* applies to node I, etc.

If *Lab* = PRES and *KVAL* = 2, *VAL1* is the imaginary pressure, which is used only by SURF153 or SURF154 in full harmonic response analyses (**HROPT**,FULL), or by a mode superposition harmonic response analysis (**HROPT**,MSUP) if the mode extraction method is Block Lanczos (**MODOPT**,LANB).

If *Lab* = CONV, *KVAL* = 0 or 1, and *VAL1* = -*N*, the film coefficient is assumed to be a function of temperature and is determined from the HF property table for material *N* [**MP**]. The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user defined for some elements.

If *Lab* = PORT, *VAL1* is a port number representing a waveguide port. The port number must be an integer between 1 and 6.

If *Lab* = RDSF, *KVAL* = 0 or 1, and *VAL1* = -*N*, the emissivity is assumed to be a function of the temperature, and is determined from the EMISS property table for material *N* (**MP**). The material *N* does not need to correlate with the underlying solid thermal elements. If *Lab* = RDSF, *KVAL* = 2, and *VAL1* is negative, radiation direction is reversed and will occur inside the element for the flagged radiation surfaces. Negative value of enclosure number is applicable for FLUID141 and FLUID142 elements, to model radiation occurring between surfaces inside the fluid domain.

If *Lab* = VFRC, *VAL1* is the boundary value or the wetting status, as defined by *KVAL* above. If *Lab* = VFRC and *KVAL* = 2, a *VAL1* setting of 1 indicates a wetted boundary. At a wetted boundary, the fluid upstream keeps the associated elements full.

If *Lab* = FSIN, *VAL1* is the interface number. *KVAL* is not used.

VAL2, *VAL3*, *VAL4*
Surface load values at the 2nd, 3rd, and 4th nodes (if any) of the face. If all three values are blank, all three default to *VAL1*, giving a constant load. Zero or other blank values are used as zero. To specify a table (*Lab* = CONV), enclose the table name in percent signs (%), e.g., %*tabname*%. Use the ***DIM** command to define a table.

Notes

Specifies surface loads on selected elements.

Caution: You cannot use the **SFE** command with the INFIN110 or INFIN111 elements without prior knowledge of element face orientation, i.e., you must know which face is the exterior in order to flag it. Also, the surface effect elements, SURF153 and SURF154, use *LKEY* to allow the import of tangential and other loads (see SURF153 and SURF154 of the *ANSYS Elements Reference* for more details).

Tapered loads may be applied over the faces of most elements. Exceptions include elements PIPE16, PIPE17, PIPE18, PIPE20, PIPE59, and PIPE60, where tapered pressures are not recognized. Only constant pressures are supported for these elements.

For beam elements allowing lateral surface loads that may be offset from the nodes, use the **SFBEAM** command to specify the loads and offsets. See the **SF** command for an alternate surface load definition capability based upon node numbers. See the **SFGRAD** command for an alternate tapered load capability. Use the **SFCUM** command to accumulate (add) surface loads applied with **SFE**.

You can specify a table name only when using structural (PRES) and thermal (CONV (film coefficient and/or bulk temperature), HFLUX), boundary value and wetting status (VFRC), and surface emissivity and ambient temperature (RAD) surface load labels. The Volume of Fluid method (*Lab* = VFRC) is applicable only for FLUID141.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Load Vector>For Superelement
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Element Components
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Elements>Tapered
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Elements>Uniform
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Surface Rad>On Elements
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Elements
Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Elements
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Elements

Main Menu>Solution>Define Loads>Apply>Load Vector>For Superelement
Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Element Components
Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Elements
Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Elements>Tapered
Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Elements>Uniform
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Elements
Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Elements
Main Menu>Solution>Define Loads>Apply>Thermal>Surface Rad>On Elements

---

SFEDELE, *ELEM*, *LKEY*, *Lab*

Deletes surface loads from elements.

SOLUTION: FE Surface Loads

MP ME ST <> <> PR EM <> FL PP ED

ELEM
Element to which surface load deletion applies. If ALL, delete load from all selected elements [**ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

LKEY
Load key associated with surface load (defaults to 1). If ALL, delete surface loads for all load keys.

Lab
Valid surface load label. If ALL, use all appropriate labels. See the **SFE** command for labels.

Notes

Deletes surface loads from selected elements. See the **SFDELE** command for an alternate surface load deletion capability based upon selected nodes.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Surface Ld>On All Elems
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppImped_E>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Volume Fract>Bound Loads>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Load Vector>For Superelement
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppMCI>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Element Components
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Surface Rad>On Elements
Main Menu>Solution>Define Loads>Delete>All Load Data>All Surface Ld>On All Elems
Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppImped_E>On Elements
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Elements
Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Elements
Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Elements
Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Elements
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Elements
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Volume Fract>Bound Loads>On Elements
Main Menu>Solution>Define Loads>Delete>Load Vector>For Superelement
Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Elements
Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppMCI>On Elements
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Elements
Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Element Components
Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Elements
Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Elements
Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Elements
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Elements
Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Elements
Main Menu>Solution>Define Loads>Delete>Thermal>Surface Rad>On Elements

SFELIST, *ELEM*, *Lab*

Lists the surface loads for elements.

SOLUTION: FE Surface Loads

MP ME ST <> <> PR EM <> FL PP ED

ELEM

Element at which surface load is to be listed. If ALL (or blank), list loads for all selected elements [**ESEL**]. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

Lab

Valid surface load label. If ALL (or blank), use all appropriate labels. Structural label: PRES (pressure). Thermal labels: CONV (convection); HFLUX (heat flux); RAD (radiation); RDSF (surface-to-surface radiation). Fluid labels: FSI (fluid-structure interaction flag); VFRC (volume fraction for VOF method); IMPD (impedance). Magnetic label: MXWF (Maxwell force flag). Substructure label: SELV (Load vector number). Electric labels: CHRGS (surface charge density); MXWF (Maxwell force flag). Infinite element label: INF (Exterior surface flag for INFIN110 and INFIN111). High-frequency electromagnetic labels: PORT (waveguide port number); SHLD (surface shielding properties). Thermal labels CONV and HFLUX are mutually exclusive. If *Lab* = SFI, only the fluid elements must be selected for the flag to be applied.

Notes

The surface loads listed correspond to the current database values. The database is not updated for surface loads in POST1. Surface loads specified in tabular form, however, do list their values corresponding to the current results set in POST1. This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Surface Loads>On All Elements
Utility Menu>List>Loads>Surface Loads>On Picked Elems

SFFUN, *Lab*, *Par*, *Par2*

Specifies a varying surface load.

SOLUTION: FE Surface Loads

MP ME ST <> <> PR EM <> FL PP ED

Lab
Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *ANSYS Elements Reference*. Structural label: PRES (pressure). Thermal labels: CONV (convection); HFLUX (heat flux). Electric labels: CHRGS (Surface Charge density). Issue **SFFUN**,STATUS to list current command settings. Thermal labels CONV and HFLUX are mutually exclusive.

Par
Parameter containing list of surface load values. If *Lab* = CONV, values are typically the film coefficients and *Par2* values (below) are typically the bulk temperatures.

Par2
Parameter containing list of second surface load values (if any). If *Lab* = CONV, the *Par2* values are typically the bulk temperatures. *Par2* is not used for other surface load labels.

Notes

Specifies a surface load "function" to be used when the **SF** or **SFE** command is issued. The function is supplied through an array parameter vector which contains nodal surface load values. Node numbers are implied from the sequential location in the array parameter. For example, a value in location 11 applies to node 11. The element faces are determined from the implied list of nodes when the **SF** or **SFE** command is issued. Zero values should be supplied for nodes that have no load. A tapered load value may be applied over the element face. These loads are in addition to any loads that are also specified with the **SF** or **SFE** commands. Issue **SFFUN** (with blank remaining fields) to remove this specification. Issue **SFFUN**,STATUS to list current settings.

Starting array element numbers must be defined for each array parameter vector. For example, **SFFUN**,CONV,A(1,1),A(1,2) reads the first and second columns of array A (starting with the first array element of each column) and associates the values with the nodes. Operations continue on successive column array elements until the end of the column.

SFFUN does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>For Surface Ld>Node Function
Main Menu>Solution>Define Loads>Settings>For Surface Ld>Node Function

---

SFGRAD, *Lab*, *SLKCN*, *Sldir*, *SLZER*, *SLOPE*

Specifies a gradient (slope) for surface loads.

SOLUTION: FE Surface Loads

MP ME ST <> <> PR EM <> FL PP ED

Lab
Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *ANSYS Elements Reference*. Structural labels: PRES (pressure). Thermal labels: CONV (convection (bulk temperatures only)); HFLUX (heat flux). Electric labels: CHRGS (surface charge density). Thermal labels CONV and HFLUX are mutually exclusive.

SLKCN
Reference number of slope coordinate system (used with *Sldir* and *SLZER* to determine COORD). Defaults to 0 (the global Cartesian coordinate system).

Sldir
Slope direction in coordinate system *SLKCN*:

X
Slope is along X direction (default). Interpreted as R direction for non-Cartesian coordinate systems.

Y
Slope is along Y direction. Interpreted as θ direction for non-Cartesian coordinate systems.

Z
Slope is along Z direction. Interpreted as Φ direction for spherical or toroidal coordinate systems.

SLZER
Coordinate location (degrees for angular input) where slope contribution is zero (CVALUE = VALUE). Allows the slope contribution to be shifted along the slope direction. For angular input, *SLZER* should be between ±180° if the singularity [**CSCIR**] is at 180° and should be between 0° and 360° if the singularity is at 0°.

SLOPE
Slope value (load per unit length or per degree).

Notes

Specifies a gradient (slope) for surface loads. All surface loads issued with the **SF**, **SFE**, **SFL**, or **SFA** commands while this specification is active will have this gradient applied (for convections, only the bulk temperature will be affected). The load value, CVALUE, calculated at each node is:

CVALUE = VALUE + (*SLOPE* X (COORD-*SLZER*))

where VALUE is the load value specified on the subsequent **SF**, **SFE**, **SFL**, or **SFA** commands and COORD is the coordinate value (in the *Sldir* direction of coordinate system *SLKCN*) of the node. Only one **SFGRAD** specification may be active at a time (repeated use of this command replaces the previous specification with the new specification). Issue **SFGRAD** (with blank fields) to remove the specification. Issue **SFGRAD**,STAT to show the current command status. The **SFGRAD** specification (if active) is removed when the **LSREAD** (if any) command is issued.

For elements PIPE16, PIPE17, PIPE18, PIPE20, and PIPE60, tapered pressures are not recognized. Only constant pressures are supported for these elements.

SFGRAD does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>For Surface Ld>Gradient
Main Menu>Solution>Define Loads>Settings>For Surface Ld>Gradient

---

SFL, *LINE*, *Lab*, *VALI*, *VALJ*, *VAL2I*, *VAL2J*

Specifies surface loads on lines of an area.

SOLUTION: Solid Surface Loads

MP ME ST <> <> PR EM <> <> PP ED

LINE

Line to which surface load applies. If ALL, apply load to all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *LINE*.

Lab

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *ANSYS Elements Reference*. Structural label: PRES (pressure). Thermal labels: CONV (convection); HFLUX (heat flux); RAD (radiation); RDSF (surface-to-surface radiation). Fluid labels: FSI (fluid-structure interaction flag); VFRC (volume fraction for VOF method); IMPD (impedance). Magnetic label: MXWF (Maxwell force flag). Electric labels: MXWF (electrostatic force flag), CHRGS (surface charge density). Infinite element label: INF (Exterior surface flag for INFIN110 and INFIN111). Field-Surface Interface label: FSIN (field-surface interface number).

> *Note* — Thermal labels CONV and HFLUX are mutually exclusive. For a fluid-solid interaction analysis, apply the field-surface interface flag (Label = FSIN) twice: once for the fluid side (FLUID141 or FLUID142 elements) and once for the solid side. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the FLUID129 or FLUID130 elements. For more information about these analyses and labels, see the *ANSYS Coupled-Field Analysis Guide* and the *ANSYS Elements Reference*.

VALI, *VALJ*

Surface load values at the first keypoint (*VALI*) and at the second keypoint (*VALJ*) of the line, or table name for specifying tabular boundary conditions. If *VALJ* is blank, it defaults to *VALI*. If *VALJ* is zero, a zero is used. If *Lab* = CONV, *VALI* and *VALJ* are the film coefficients and *VAL2I* and *VAL2J* are the bulk temperatures. To specify a table, enclose the table name in percent signs (%), e.g., %*tabname*%. Use the ***DIM** command to define a table. If *Lab* = CONV and *VALI* = -*N*, the film coefficient may be a function of temperature and is determined from the HF property table for material *N* [**MP**]. If *Lab* = RAD, *VALI* and *VALJ* values are surface emissivities and *VAL2I* and *VAL2J* are ambient temperatures. The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user defined for some elements. If *Lab* = RDSF, *VALI* is the emissivity value; the following condition apply: If *VALI* = -*N*, the emissivity may be a function of the temperature and is determined from the EMISS property table for material *N* [**MP**]. If *Lab* = VFRC, *VALI* is the boundary value (defaults to 1). If *Lab* = FSIN, *VALI* is the interface number. *VALJ*, *VAL2I*, and *VAL2J* are not used.

VAL2I, *VAL2J*

Second surface load values (if any). If *Lab* = CONV, *VAL2I* and *VAL2J* are the bulk temperatures. If *Lab* = RAD, *VAL2I* and *VAL2J* are the ambient temperatures. If *Lab* = RDSF, *VAL2I* is the enclosure number. Radiation will occur between surfaces flagged with the same enclosure numbers. If the enclosure is open, radiation will occur to the ambient. Negative value of enclosure number is applicable for FLUID141 and FLUID142

elements, to model radiation occurring between surfaces inside the fluid domain. *VAL2I* and *VAL2J* are not used for other surface load labels. If *VAL2J* is blank, it defaults to *VAL2I*. If *VAL2J* is zero, a zero is used. To specify a table (*Lab* = CONV), enclose the table name in percent signs (%), e.g., %*tabname*%. Use the ***DIM** command to define a table. If *Lab* = VFRC, *VAL2I* is the wetting status value (defaults to 0 or non-wetting). If *Lab* = VFRC, a *VAL2I* setting of 1 indicates a wetted boundary. At a wetted boundary, the fluid upstream keeps the associated elements full.

Notes

Specifies surface loads on the selected lines of area regions. The lines represent either the edges of area elements or axisymmetric shell elements themselves. Surface loads may be transferred from lines to elements with the **SFTRAN** or **SBCTRAN** commands. See the **SFE** command for a description of surface loads. Loads input on this command may be tapered. See the **SFGRAD** command for an alternate tapered load capability.

You can specify a table name only when using structural (PRES) and thermal (CONV (film coefficient and/or bulk temperature), HFLUX), boundary value and wetting status (VFRC), and surface emissivity and ambient temperature (RAD) surface load labels. *VALJ* and *VAL2J* are ignored for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppInfinite>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppMaxwell>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Surface Rad>On Lines
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Lines
Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppInfinite>On Lines
Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppMaxwell>On Lines
Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Lines
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Lines
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Lines
Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Lines
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Lines
Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Lines
Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Lines
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Lines
Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Lines
Main Menu>Solution>Define Loads>Apply>Thermal>Surface Rad>On Lines

SFLDELE, *LINE*, *Lab*

Deletes surface loads from lines.

SOLUTION: Solid Surface Loads

MP ME ST <> <> PR EM <> <> PP ED

LINE
Line to which surface load deletion applies. If ALL, delete load from all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *LINE*.

Lab
Valid surface load label. If ALL, use all appropriate labels. See the **SFL** command for labels.

Notes

Deletes surface loads (and all corresponding finite element loads) from selected lines.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Surface Ld>On All Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/CFD>Volume Fract>Bound Loads>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Surface Rad>On Lines
Main Menu>Solution>Define Loads>Delete>All Load Data>All Surface Ld>On All Lines
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Lines
Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Lines
Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Lines
Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Lines
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Lines
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Lines
Main Menu>Solution>Define Loads>Delete>Fluid/CFD>Volume Fract>Bound Loads>On Lines
Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Lines
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Lines
Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Lines
Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Lines
Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Lines

Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Lines
Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Lines
Main Menu>Solution>Define Loads>Delete>Thermal>Surface Rad>On Lines

---

SFLIST, *NODE*, *Lab*

Lists surface loads.

SOLUTION: FE Surface Loads

MP ME ST <> <> PR EM <> FL PP ED

NODE

Node at which surface load is to be listed. If ALL (or blank), list for all selected nodes [**NSEL**]. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NODE*.

Lab

Valid surface load label. If ALL (or blank), use all appropriate labels. Structural labels: PRES (pressure). Thermal labels: CONV (convection); HFLUX (heat flux); RAD (radiation); RDSF (surface-to-surface radiation). Fluid labels: FSI (fluid-structure interaction flag); IMPD (impedance). Magnetic labels: MXWF (Maxwell force flag); MCI (magnetic circuit interface). Electric label: CHRGS (surface charge density); MXWF (Maxwell force flag). High-frequency electromagnetic labels: PORT (waveguide port number); SHLD (surface shielding properties). Infinite element label: INF (exterior surface flag for INFIN110 and INFIN111). If *Lab* = FSI, only the fluid elements must be selected for the flag to be applied.

Notes

Lists the surface loads as applied with the **SF** command. Loads are listed only for the specified nodes on external faces of selected area and volume elements. Use **SFELIST** for line elements. The surface loads listed correspond to the current database values. The database is not updated for surface loads in POST1. Surface loads specified in tabular form, however, do list their values corresponding to the current results set in POST1.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Surface Loads>On All Nodes
Utility Menu>List>Loads>Surface Loads>On Picked Nodes

SFLLIST, *LINE*, *Lab*

Lists the surface loads for lines.

SOLUTION: Solid Surface Loads

MP ME ST <> <> PR EM <> <> PP ED

LINE

Line at which surface load is to be listed. If ALL (or blank), list for all selected lines [**LSEL**]. If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *LINE*.

Lab
Valid surface load label. If ALL (or blank), use all appropriate labels. See the **SFL** command for labels.

Notes

Lists the surface loads for the specified line.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Surface Loads>On All Lines
Utility Menu>List>Loads>Surface Loads>On Picked Lines

SFSCALE, *Lab*, *FACT*, *FACT2*

Scales surface loads on elements.

SOLUTION: FE Surface Loads

MP ME ST <> <> PR <> <> FL PP ED

Lab
Valid surface load label. If ALL, use all appropriate labels. Structural label: PRES (pressure). Thermal labels: CONV (convection); HFLUX (heat flux). Substructure label: SELV (Load vector number). Electric labels: CHRGS (surface charge density). Thermal labels CONV and HFLUX are mutually exclusive.

FACT
Scale factor for the first surface load value. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

FACT2
Scale factor for the second surface load value. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

Notes

Scales surface loads (pressure, convection, etc.) in the database on the selected elements. Surface loads are applied with the **SF**, **SFE**, or **SFBEAM** commands. Issue the **SFELIST** command to list the surface loads. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

Note — Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

SFSCALE does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Surface Loads
Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Surface Loads

---

SFTRAN

Transfer the solid model surface loads to the finite element model.

SOLUTION: Solid Surface Loads

MP ME ST <> <> PR EM <> <> PP ED

Notes

Surface loads are transferred only from selected lines and areas to all selected elements. The **SFTRAN** operation is also done if the **SBCTRAN** command is issued or automatically done upon initiation of the solution calculations [**SOLVE**].

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Surface Loads
Main Menu>Solution>Define Loads>Operate>Transfer to FE>Surface Loads

---

/SHADE, *WN*, *Type*

Defines the type of surface shading used with Z-buffering.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

Type
Shading type:

FACET or 0
Facet shading (one color per area face) (default).

GOURAUD or 1
Gouraud smooth shading (smooth variation of color based on interpolated vertex colors).

PHONG or 2
Phong smooth shading (smooth variation of color based on interpolated vertex normals).

Command Default

Facet shading.

Notes

Defines the type of surface shading used on area, volume, and PowerGraphics [**/GRAPHICS**,POWER] displays when software Z-buffering is enabled [**/TYPE**]. This command is only functional for 2-D display devices.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

SHELL, *Loc*

Selects a shell element or shell layer location for results output.

POST1: Controls
POST26: Controls

MP ME ST <> <> PR <> <> <> PP ED

Loc

Location within shell element (or layer) to obtain stress results:

TOP
: Top of shell element (or layer) (default).

MID
: Middle of shell element (or layer). The default method averages the TOP and BOT values to obtain a mid value. Setting KEYOPT(8) = 2 for SHELL93, SHELL181, SHELL208, and SHELL209, or KEYOPT(11) = 2 for SHELL63 uses MID results obtained directly from the results file.

BOT
: Bottom of shell element (or layer).

Command Default

Shell element (or layer) top location.

Notes

Selects the location within a shell element (or a shell layer) for results output (nodal stresses, strains, etc.). Applies to POST1 selects, sorts, and output [**NSEL**, **NSORT**, **PRNSOL**, **PLNSOL**, **PRPATH**, **PLPATH**, etc.], and is used for storage with the POST26 **ESOL** command. For example, **SHELL**,TOP causes item S of the POST1 **PRNSOL** command or the POST26 **ESOL** command to be the stresses at the top of the shell elements. For layered shell elements, use the **LAYER** (POST1) or **LAYERP26** (POST26) command to select the layer. The **SHELL** command does not apply to the layered thermal shell elements, SHELL131 and SHELL132. For PowerGraphics [**/GRAPHICS**,POWER], the **SHELL**,MID command affects both the printed output and the displayed results, while the **SHELL** (TOP or BOT) command prints and displays both the top and bottom layers simultaneously.

In POST26, the **ESOL** data stored is based on the active **SHELL** specification at the time the data is stored. To store data at various specifications (for example, stresses at the top and bottom locations), issue a **STORE** command before each new specification.

Menu Paths

Main Menu>General Postproc>Options for Outp
Main Menu>TimeHist Postpro>Define Variables
Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables
Utility Menu>List>Results>Options

/SHOW, *Fname*, *Ext*, *VECT*, *NCPL*

Specifies the device and other parameters for graphics displays.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Fname

Device name, filename, or keyword, as listed below:

<*device name*>

Any valid graphics display device name (e.g., X11, 3-D etc.). Defaults to X11 for most systems. See Getting Started with Graphics in the *ANSYS Basic Analysis Guide* for details. A device name must be defined before activating the Graphical User Interface (GUI). Once the GUI is activated, the device name cannot be changed for that ANSYS session, except for switching between X11 and X11C.

<*filename*>

Name of graphics file to which graphics displays are to be diverted (248 characters maximum). Should not be the same as a valid device name or any other *Fname* option. Plots are written to the file ***Filename.Ext*** (or just ***Filename.Ext*** if *Ext* is left blank) in the working directory. If this file already exists, it will be overwritten unless the command is being reissued in the same ANSYS session.

TERM

Graphics displays are switched back to the last-specified device name.

CLOSE

This option purges the graphics file buffer. The CLOSE option should be issued any time you are changing graphics devices or file output types during an ANSYS session. Graphics displays are switched back to the last-specified device name, and any open graphics files are closed. The CLOSE option is similar to the TERM option, however, with the CLOSE option, another process, such as the DISPLAY program, can access the data in the graphics file. The CLOSE option causes graphics file buffers to be flushed to the graphics file.

FILE

Graphics displays are switched back to the last-specified file name.

OFF

Graphics display requests are ignored.

(blank)

If blank in interactive mode, graphics will be displayed on screen as requested by display commands (no file written); If blank in batch mode, graphics data will be written to **Jobname.GRPH**.

PSCR

Creates PostScript graphic files that are named **Jobname*nnn*.eps**, where *nnn* is a numeric value that is incremented by one as each additional file is created; that is, **Jobname000.eps**, **Jobname001.eps**, **Jobname002.eps**, and so on . (See the **PSCR** command for options.) Ignores the *Ext* and *NCPL* fields.

HPGL

Creates Hewlett-Packard Graphics Language files that are named **Jobname*nnn*.hpgl**, where *nnn* is a numeric value that is incremented by one as each additional file is created; that is, **Jobname000.hpgl**, **Jobname001.hpgl**, **Jobname002.hpgl**, and so on. (See the **HPGL** command for options.) Ignores the *Ext* and *NCPL* fields.

HPGL2

Creates Hewlett-Packard Graphics Language files that are named **Jobname*nnn*.hpgl**, where *nn* is a numeric value that is incremented by one as each additional file is created; that is, **Jobname000.hpgl**,

Jobname001.hpgl, **Jobname02.hpgl**, and so on. The *HPGL2* files have enhanced color. (See the **HPGL** command for options.) Ignores the *Ext* field.

JPEG
: Creates JPEG files that are named **Jobname*nnn*.jpg**, where *nnn* is a numeric value that is incremented by one as each additional file is created; that is, **Jobname000.jpg**, **Jobname001.jpg**, **Jobname002.jpg**, and so on. Ignores the *Ext* field.

TIFF
: Creates tagged image format files that are named **Jobname*nnn*.tif**, where *nnn* is a numeric value that is incremented by one as each additional file is created; that is, **Jobname000.tif**, **Jobname001.tif**, **Jobname002.tif**, and so on. This value for the *Fname* argument ignores the *Ext* field. (See the **TIFF** command for options.)

PNG
: Creates PNG (Portable Network Graphics) files that are named **Jobname*nnn*.png**, where *nnn* is a numeric value that is incremented by one as each additional file is created; that is, **Jobname000.png**, **Jobname001.png**, **Jobname002.png**, and so on. This value for the *Fname* argument ignores the *Ext* field. (See the **PNGR** command for options.)

VRML
: Creates Virtual Reality Meta Language files named **Jobname000.wrl** that can be displayed on 3-D Internet web browsers. Ignores the *Ext* and *NCPL* fields.

Ext
: Filename extension (8 character maximum).

VECT
: Specifies raster or vector display mode. This affects area, volume, and element displays, as well as geometric results displays such as contour plots. See the **/DEVICE** command for an alternate way to toggle between raster and vector mode. Changing *VECT* also resets the **/TYPE** command to its default.

0
: Raster display (color filled entities; default)

1
: Vector display (outlined entities; i.e., "wireframe")

NCPL
: Sets the number of color planes (4 to 8). Default is device-dependent. *NCPL* is not supported by all graphics devices.

Command Default

For interactive runs, display is shown on the screen; for batch runs, display commands are ignored (graphics file not written).

Notes

Specifies the device to be used for graphics displays, and specifies other graphics display parameters. Display may be shown at the time of generation (for interactive runs at a graphics display terminal) or diverted to a file for later processing with the DISPLAY program. Issue **/PSTATUS** for display status.

Batch runs do not have access to the fonts available on your system. The Courier and Helvetica font files used for JPEG, PNG and TIFF batch output are copyrighted by Adobe Systems Inc. and Digital Equipment Corp. Per-

mission to use these trademarks is hereby granted only in association with the images described above. Batch run JPEG output is produced at the default quality index value of 75, unless specified otherwise.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Device Options
Utility Menu>PlotCtrls>Redirect Plots>To GRPH File
Utility Menu>PlotCtrls>Redirect Plots>To HPGL File
Utility Menu>PlotCtrls>Redirect Plots>To HPGL2 File
Utility Menu>PlotCtrls>Redirect Plots>To PSCR File
Utility Menu>PlotCtrls>Redirect Plots>To Screen

/SHOWDISP, *Dname*, --, --, *NCPL*

Defines the display driver name.

DISPLAY: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Dname
Valid driver name (see Getting Started with Graphics in the *ANSYS Basic Analysis Guide* for details):

<device name>
Any linked terminal driver (such as X11, TEKTRONIX, etc.)

HPGL
Hewlett-Packard Graphics Language

HPGL2
Hewlett-Packard Graphics Language with enhanced color. (See the **HPGL** command for options.) Ignores the *NCPL* field.

INTERLEAF
Interleaf ASCII Format, OPS Version 5.0

POSTSCRIPT
PostScript, Version 1.0 Minimally Conforming

DUMP
ASCII Text Dump

--, --
Unused fields.

NCPL
Number of color planes (4 to 8). Default is device-dependent.

Menu Paths

It is part of the DISPLAY command.

SHPP, *Lab*, *VALUE1*, *VALUE2*

Controls element shape checking.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

Lab

Shape checking option. (When *Lab* = WARN, STATUS, SUMMARY, or DEFAULT, the remaining arguments are ignored.)

ON

Activates element shape checking. New elements, regardless of how they are created, are tested against existing warning and error limits. (The existing limits may be the default limits, or previously modified limits.) Elements that violate error limits produce *error* messages and either (a) cause a meshing failure, or (b) for element creation or storage other than **AMESH** or **VMESH**, *are not stored.* Elements that violate warning limits produce *warning* messages. If shape checking was previously turned off [**SHPP**,OFF] and you turn it on, existing elements are marked as untested; use the **CHECK** command to retest them. With this option, you may also specify a value for *VALUE1* to turn individual shape tests on. If you do not specify a value for *VALUE1*, all shape tests are turned on.

WARN

Activates element shape checking; however, in contrast to **SHPP**,ON, elements that violate error limits do not cause either a meshing or element storage failure. Instead, they produce *warning* messages to notify you that error limits have been violated. This option does not alter current shape parameter limits. Since the default shape parameter error limits are set to allow almost any usable element, the elements this option allows, which would otherwise be forbidden, are likely to be *very poorly shaped*.

OFF

Deactivates element shape checking. This setting does not alter current shape parameter limits. Use of this option is risky, since poorly shaped elements can lead to analysis results that are less accurate than would otherwise be expected for a given mesh density. With this option, you may also specify a value for *VALUE1* to turn individual shape tests off. If you do not specify a value for *VALUE1*, all element shape tests are turned off.

SILENT

Determines whether element shape checking runs in silent mode. In silent mode, ANSYS checks elements without issuing warnings, *with the exception of* the generic warnings that it issues at solution. With this option, you must also specify a value for *VALUE1* (During the execution of certain commands, ANSYS automatically runs element shape checking in silent mode, then internally summarizes the shape test results for all of the new or modified elements. ANSYS does this when it executes any of the following commands: **AGEN**, **AMESH**, **AREFINE**, **ARSYM**, **ATRAN**, **CDREAD**, **EGEN**, **ENGEN**, **ENSYM**, **EREAD**, **EREFINE**, **ESYM**, **ET**, **FVMESH**, **KREFINE**, **LREFINE**, **NREFINE**, **TIMP**, **VEXT**, **VGEN**, **VIMP**, **VMESH**, **VOFFST**, **VROTAT**, **VSWEEP**, **VSYMM**, and **VTRAN**.)

STATUS

Lists the shape parameter limits currently in effect, along with status information about element shape checking (for example, whether any individual shape tests are off, whether any of the shape parameter limits have been modified, and so on).

SUMMARY

Lists a summary of element shape test results for all selected elements.

DEFAULT

Resets element shape parameter limits to their default values. Also, if any individual tests were turned off, turns them back on. (The **SHPP**,DEFAULT command may be useful if any parameter limits were previously altered by using the MODIFY option.)

OBJECT

Determines whether element shape test results data is *stored in memory*. When this option is turned on, an "object" is created for storing test results in memory. When this option is turned off, no object is created and no data is stored; thus, any operation that requires shape parameters for an existing element (such as use of the **CHECK** command) causes the shape parameters to be recomputed. (Note the distinction between storing the data in memory and storing it in the database; regardless of whether this option is turned on or off, no element shape test results data will be stored in the database. The element shape parameter object is deleted automatically before any solution.) This setting is independent of shape checking status, with one exception--if shape checking is turned off [**SHPP**,OFF], the object is not created. Keep in mind that recomputing shape parameters is more computationally expensive than retrieving them from the object. With this option, you must also specify a value for the *VALUE1* argument; the *VALUE2* argument is ignored.

LSTET

Determines, for Jacobian ratio tests, whether sampling is done at integration points (DesignSpace product method), or at corner nodes. When this option is turned on, sampling is done at integration points, and the default limits for h-element Jacobian ratios are a warning tolerance of 10 and an error tolerance of 40. When this option is turned off, sampling is done at corner nodes, and the corresponding default limits are a warning tolerance of 30 and an error tolerance of 1000. Sampling at the integration points (option on) results in a lower Jacobian ratio, but that ratio is also subjected to a more restrictive error limit. Some elements that have passed the integration point sampling criterion, have failed the corner mode sampling criterion. Because of this, use integration point sampling only for simple linear analyses. For other types of analyses (e.g., nonlinear, electromagnetic), use sampling at corner nodes, which is the more conservative approach. With this option, you must also specify a value for the *VALUE1* argument; the *VALUE2* argument is ignored.

MODIFY

Indicates that you want to respecify a shape parameter limit. With this option, you must also specify values for the *VALUE1* and *VALUE2* arguments.

FLAT

Determines the warning and error limits used to test elements that may exhibit nonzero/nonconstant Z coordinates. With this option, you must also specify values for the *VALUE1* and/or *VALUE2* arguments.

VALUE1

Valid for the ON, OFF, SILENT, OBJECT, LSTET, MODIFY, and FLAT options only. When *Lab* = ON or OFF, use *VALUE1* to individually control (that is, turn off or turn on) specific element shape tests. Thus, *VALUE1* can be ANGD (SHELL28 corner angle deviation tests), ASPECT (aspect ratio tests), PARAL (deviation from parallelism of opposite edges tests), MAXANG (maximum corner angle tests), JACRAT (Jacobian ratio tests), WARP (warping factor tests), or ALL (all tests). When *Lab* = SILENT, *VALUE1* can be ON (to turn silent mode on) or OFF (to turn silent mode off). When *Lab* = OBJECT, *VALUE1* can be either 1, YES, or ON to turn on storage of element shape test data (the default); or it can be 0, NO, or OFF to turn off storage of element shape test data (delete the data and recompute as necessary). When *Lab* = LSTET, *VALUE1* can be either 1, YES, or ON to choose Jacobian sampling at integration points; or it can be 0, NO, or OFF to choose Jacobian sampling at nodes (the default). When *Lab* = MODIFY, *VALUE1* is the numeric location (within the shape parameter limit array) of the shape parameter limit to be modified. Locations are identified in the element shape checking status listing [**SHPP**,STATUS]. For more information, see the examples in the **Notes** section. When *Lab* = FLAT, *VALUE1* is the warning limit for XY element constant Z sets performed at **CHECK** or **SOLVE**. The default is 1.0e-8.

VALUE2

Valid for the MODIFY and FLAT options only. When *Lab* = MODIFY, specifies the new limit for the shape parameter that is in the location indicated by the *VALUE1* argument. See the examples in the **Notes** section. When *Lab* = FLAT, *VALUE2* is the error limit. The default is 1.0e-2.

Command Default

All shape checking tests are on [**SHPP**,ON,ALL] with default shape parameter limits. Silent mode is off. Memory object storage of element shape parameters is on.

Notes

The following examples illustrate how to use the **SHPP**,MODIFY,*VALUE1*,*VALUE2* command to respecify shape parameter limits. Assume that you issued the **SHPP**,STATUS command, and you received the output below:

```
 ASPECT RATIO (EXCEPT FLOTRAN OR EMAG)

    QUAD OR TRIANGLE ELEMENT OR FACE
         WARNING TOLERANCE ( 1) =   20.00000
         ERROR TOLERANCE   ( 2) =  1000000.
                    .
                    .
                    .
 MAXIMUM CORNER ANGLE IN DEGREES (EXCEPT FLOTRAN OR EMAG)
    TRIANGLE ELEMENT OR FACE
         WARNING TOLERANCE (15) =   165.0000
         ERROR TOLERANCE   (16) =   179.9000
```

Notice that in the sample output, the *warning* tolerance for aspect ratios is set to 20. Now assume that you want to "loosen" this shape parameter limit so that it is less restrictive. To allow elements with aspect ratios of up to 500 without causing warning messages, you would issue this command:

```
SHPP,MODIFY,1,500
```

Also notice that each shape parameter's numeric location within the shape parameter limit array appears in the sample output within parentheses. For example, the numeric location of the aspect ratio shape parameter (for warning tolerance) is 1, which is why "1" is specified for the *VALUE1* argument in the example command above.

Now notice that the sample output indicates that any triangle element with an internal angle that is greater than 179.9 degrees will produce an *error* message. Suppose that you want to "tighten" this shape parameter limit, so that it is more restrictive. To cause any triangle or tetrahedron with an internal angle greater than 170 degrees to produce an error message, you would issue this command:

```
SHPP,MODIFY,16,170
```

Warning: The existence of badly shaped elements in a model may lead to certain computational errors that can cause your system to abort during ANSYS solution. Therefore, you run the risk of a system abort during solution any time that you turn element shape checking off entirely, run shape checking in warning-only mode, turn off individual shape checks, or loosen shape parameter limits.

Changing any shape parameter limit marks all existing elements as untested; use the **CHECK** command to retest them.

Since the shape parameter limit array was completely reorganized at ANSYS 5.4, you should revise any input files created prior to 5.4 that contain limit changes so that they reflect the reorganized data structure.

For more information about element shape checking, see Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Checking Ctrls>Shape Checking
Main Menu>Preprocessor>Checking Ctrls>Toggle Checks

/SHRINK, *RATIO*

Shrinks elements, lines, areas, and volumes for display clarity.

GRAPHICS: Scaling

MP ME ST DY <> PR EM <> FL PP ED

RATIO

Shrinkage ratio (input as a decimal (0.0 to 0.5)). Defaults to 0.0 (no shrinkage). Values greater than 0.5 default to 0.1 (10% shrinkage).

Command Default

Full size entities.

Notes

Shrinks the elements, lines, areas, and volumes so that adjacent entities are separated for clarity. This command is not valid with p-elements. Portions of this command are not supported by PowerGraphics [**/GRAPHICS**,POWER].

If only the common lines of non-coplanar faces are drawn (as per the **/EDGE** command), then this command is ignored.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

SHSD, *RID*, *Action*

Creates or deletes shell-solid interface to be used in shell-to-solid assemblies.

PREP7: Elements

MP ME <> <> <> PR <> <> <> PP ED

RID

The real constant set ID that identifies the contact pair on which a shell-to-solid assembly is defined. If ALL, all selected contact pairs will be considered for assembly.

Action

CREATE
(default) Builds new shell and contact elements to be used in shell-solid assemblies. New elements are stored as internally-created components.

DELETE
Deletes the nodes and elements created during a previous execution of **SHSD**,*RID*,CREATE for the real constant set identified by *RID*.

Notes

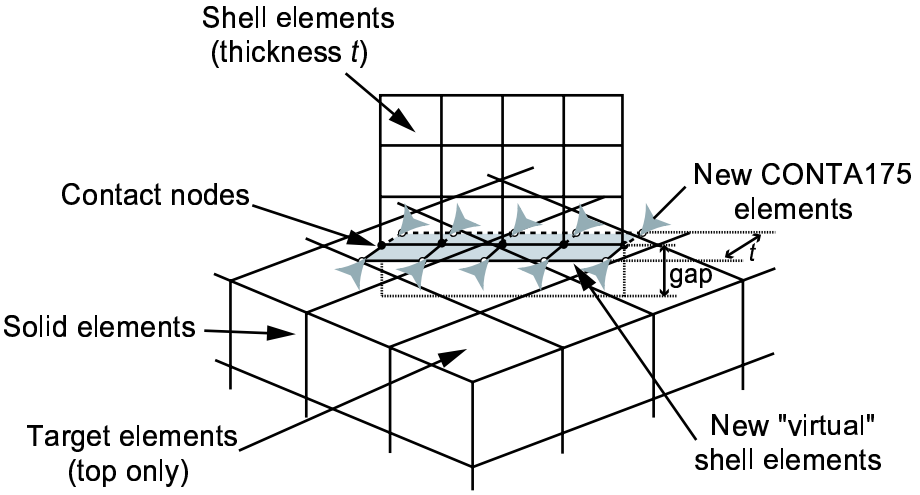
The **SHSD** command creates a shell-solid interface to be used in shell-to-solid assemblies, or deletes a previously-created shell-solid interface. "Virtual" shell elements and additional CONTA175 elements are created at the contact pair identified by *RID* when *Action* = CREATE. Set *Action* = DELETE to remove the generated nodes and elements at the contact pair identified by *RID*.

The **SHSD** command is active only when the following element KEYOPTs of associated CONTA175 and TARGE170 element types are predefined:

Element	KEYOPT	Detail
CONTA175	KEYOPT(2) = 2	MPC algorithm
	KEYOPT(12) = 5, 6	bonded contact
	KEYOPT(4) = 0	contact normal perpendicular to target surface
TARGE170	KEYOPT(5) = 1, 2	types of constraints: (1 = solid-solid, 2 = shell-shell)

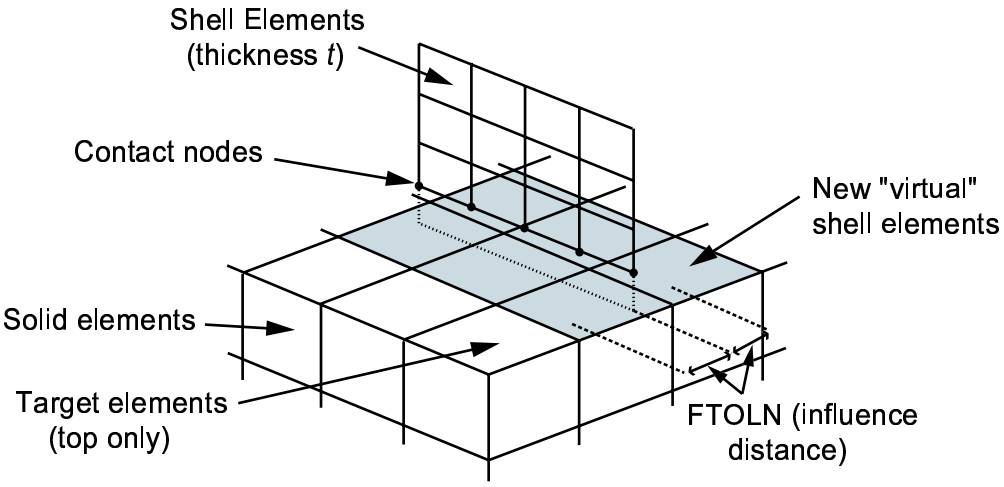
The method used to build shell and contact elements depends on the KEYOPT(5) setting of the target element type associated with the real constant set identified with the *RID* argument. If KEYOPT(5) = 1 (solid-solid constraint), the "virtual" shell elements are built perpendicularly to the pre-existing shell elements attached to the contact elements. They geometrically follow the contact interface edge and are built on both sides of this interface in such a way that each new shell element (SHELL181) has two nodes that belong to the associated pre-existing shell element in the shell edge (see "Virtual" Shell Elements Following the Contact Interface Edge). The width of the new shell elements is half the thickness of the pre-existing shell element. The CONTA175 elements are then created at each node of the "virtual" shell elements where no CONTA175 element exists. The new contact elements are identified by the same contact pair ID as the pre-existing contact elements.

"Virtual" Shell Elements Following the Contact Interface Edge



If KEYOPT(5) = 2 (shell-shell constraint), the "virtual" shell elements (SHELL181: low order; SHELL93: high order) overlap the existing high or low order target elements identified with the *RID* argument, and share their nodes. Only those target elements close enough to the contact interface (identified using the PINB real constant) are overlapped. ANSYS uses the FTOLN real constant (defaults to half the shell element thickness) to define an influence distance. The associated virtual shell elements are created only for those target elements which lie partially inside the influence distance region (see "Virtual" Shell Elements Overlapping Target Elements).

"Virtual" Shell Elements Overlapping Target Elements



For the bonded always option (KEYOPT(12) = 5), any contact node inside the pinball region (gap < PINB) is included in the KEYOPT(5) = 2 process. A relatively small PINB value may be used to prevent false contact. PINB defaults to 25% of the contact depth for small deformation analyses.

For the bonded initial option (KEYOPT(12) = 6), only those contact nodes which initially lie inside the adjustment zone (gap < ICONT) are always included in the KEYOPT(5) = 2 process. ICONT defaults to 5% of the contact depth.

For both processes, the new nodes and elements are stored in internally-named components. The internal naming convention is based on the real constant set ID specified by *RID*, as illustrated in the following table.

Nodes	SHSD_ND_*RID*

Contact Elements	SHSD_CN_*RID*
Shell Elements	SHSD_SH_*RID*

Issuing **SHSD**,*RID*,DELETE deletes components based on their generated names. *Only components whose names match the internal naming convention will be deleted.*

Caution: Do *not* rename or manually delete generated components. Use the **SHSD** command to delete generated components.

Renaming or manually deleting generated components will cause these components to be ignored when **SHSD**,*RID*,DELETE is executed and when ANSYS searches for these components to verify if **SHSD**,*RID*,CREATE can be safely executed. Manually renaming or deleting generated components and reissuing **SHSD**,*RID*,CREATE may result in erroneous generation of virtual shell or contact elements.

See Shell-solid Assembly in the *ANSYS Structural Analysis Guide* for more information.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Shell/Solid Interface

SLIST, *SFIRST*, *SLAST*, *SINC*, *Details*, *Type*

Summarizes the section properties for all defined sections in the current session of ANSYS.

PREP7: Cross Sections

MP ME ST <> <> <> <> <> <> PP ED

SFIRST
First section ID to be summarized. Defaults to first available section in the database.

SLAST
Last section ID to be summarized. Defaults to last available section in the database.

SINC
Increment of the section ID; defaults to 1.

Details
Determines the content of the summarized information for beams and shells.

BRIEF
For beams, lists only the section integrated properties (such as *Area*, *Iyy*, and *Iyz*). This option is the default.

FULL
For beams, lists the section integrated properties, as well as the section nodal coordinates, section cell connectivity information, and section cell integration point coordinates. For shells, the section stiffness (membrane, bending, membrane-bending coupling and transverse shear) are printed.

The shell section stiffness listed considers elastic behavior of materials at reference temperature only. The elements that use the section data may alter the transverse shear stiffness based on slenderness considerations (in addition to the shear correction factors shown).

44
You can use section commands with BEAM44 elements instead of defining real constants to describe the elements. Issuing **SLIST**,,,,44 lists the section integrated properties, as well as the real constant

commands that are equivalent to your section command specifications. A section made of multiple materials cannot be modeled using BEAM44.

This option does not apply to shell sections.

Type
The section type. Valid arguments are ALL (the default), BEAM, SHELL, or PRETENSION.

Notes

For BEAM188 and BEAM189 elements, the **PRSSOL** command prints section nodal and section integration point results. Stresses and strains are printed at section nodes, and plastic strains and plastic work are printed at section integration points. **PRSSOL** does not support BEAM44 elements.

By default, the command lists information concerning all sections; however, you can limit the output to only beam or pretension sections via the *Type* key.

Sample Output

Following is sample output from the **SLIST**,,,,BRIEF command for a rectangle beam section subtype:

```
LIST SECTION ID SETS     1 TO     1 BY     1

  SECTION ID NUMBER:         1
  BEAM SECTION TYPE:     Rectangle
  BEAM SECTION NAME IS:
  BEAM SECTION DATA SUMMARY:
   Area                 =  6.0000
   Iyy                  =  4.5000
   Iyz                  = 0.11281E-15
   Izz                  =  2.0000
   Warping Constant     = 0.23299
   Torsion Constant     =  4.7330
   Center of Gravity Y  =-0.30973E-16
   Center of Gravity Z  = 0.15376E-15
   Shear Center Y       =-0.22957E-13
   Shear Center Z       = 0.31281E-13

   Beam Section is offset to CENTROID of cross section
```

Menu Paths

Main Menu>Preprocessor>Sections>List Sections

SLOAD, *SECID*, *PLNLAB*, *KINIT*, *KFD*, *FDVALUE*, *LSLOAD*, *LSLOCK*

Load a pretension section.

PREP7: Cross Sections

MP ME ST <> <> <> <> <> <> PP ED

SECID
Unique section number. The number must already be assigned to a pretension section.

PLNLAB
Label representing the pretension load sequence number in the format "PL*nn*" where *nn* is an integer from 1 through 99 (for example, PL01 through PL99).

Specify a value of DELETE to delete all loads on the specified pretension section (*SECID*). In this case, the command ignores any other argument values.

KINIT
Initial action key for pretension load PL01. (This field is omitted for PL02 and up.) Three scenarios are possible:

LOCK
Constrains (connects) the cutting plane on the pretension section. This value is the default.

SLID
Unconstrains (disconnects) the cutting plane on the pretension section.

TINY
Applies a very small pretension load (0.1% of *FDVALUE*) before the desired load is established. The small load prevents convergence problems which can occur when the desired load is not established in the first load step. This value is valid only if *KFD* = FORC.

KFD
Force/Displacement key. Specifies whether *FDVALUE* is a force or a displacement:

FORC
Apply a force on the specified pretension section. This value is the default.

DISP
Apply a displacement (adjustment) on the specified pretension section.

FDVALUE
Pretension load value. If *KFD* = FORC, this value is a pretension force. If *KFD* = DISP, this value is a pretension displacement (adjustment).

LSLOAD
Load step in which to apply the *FDVALUE*.

LSLOCK
The load step in which the displacement value resulting from the pretension force is locked. This value is valid only if *KFD* = FORC.

Command Default

The default pretension load value *FDVALUE* is zero (no load). A positive value puts the pretension elements in tension.

No default exists for the *LSLOAD* applied load step value. You must specify the load step in which to apply the *FDVALUE*.

No default exists for the *LSLOCK* locked load step value. You must specify the load step in which to lock the *FDVALUE*.

Notes

The **SLOAD** command applies pretension loads to specified pretension sections (created via the **PSMESH** command). A pretension load is ramp-applied (**KBC** = 0) if it is a force (*KFD* = FORC), and step-applied (**KBC** = 1) if it is a displacement (*KFD* = DISP).

You can "lock" the load value at a specified load step. When locked, the load changes from a force to a displacement, and ANSYS applies the load as a constant displacement in all future load steps. Locking is useful when

applying additional loadings. The additional loadings alter the effect of the initial load value, but because locking transforms the load into a displacement, it preserves the initial load's effect.

The following command shows how to establish loads on a pretension section:

```
SLOAD,1,PL01,TINY,FORC,5000,1,2
```

In this example, the load is applied to pretension section **1**, and the sequence begins with the initial action key, *KINIT*, set to TINY. The next four fields set the first load: the *KFD* value FORC specifies the type of load, *FDVALUE* defines the pretension load value (5000), *LSLOAD* specifies the load step in which the force is applied (1), and the *LSLOCK* field specifies the load step in which the force is locked (2). Additional sets of four fields can be used to define additional loads.

You can use the **SLOAD** command to edit (overwrite) existing loads on a pretension section. This example changes the load on pretension section 1 (set above) to 6000:

```
SLOAD,1,PL01,,,6000,1,2
```

Unspecified values (blank fields), as shown in this example, remain unchanged from prior settings. If no prior specifications exist, then default values (*KINIT* = LOCK and *KFD* = FORC) apply.

The command can also delete all loads on a specified pretension section, as shown here:

```
SLOAD,1,DELETE
```

For a prestressed modal analysis, this command locks the pretension element:

```
SLOAD,1,PL01,LOCK,DISP,0,1,2
```

Multiple Loadings

The **SLOAD** command allows you to apply multiple loadings. You can add up to 15 loadings (PL01 through PL15), or delete loadings, for any given pretension section(s).

The following **SLOAD** commands, issued in the order shown, establish a pretension load sequence in pretension section 2 with a force of 25 in load step (LS) 2, locked in LS 3-6, a force of 50 in LS 7, locked in LS 8-11, a force of 75 in LS 12, locked in LS 13 and beyond:

```
SLOAD,2,PL01,LOCK,FORC,25,2,3

SLOAD,2,PL02,,FORC,50,7,8

SLOAD,2,PL03,,FORC,75,12,13
```

At the same time, you can issue SLOAD commands to apply loads on *other* pretension sections. For example, in addition to the commands listed above, you could issue the following command to apply a load on pretension section 3:

```
SLOAD,3,PL01,LOCK,FORC,25,3,4
```

Using the Pretension Section Loads GUI

Any addition or deletion of a loading applies to the selected sections only. ANSYS does not apply or delete a load until you click on the **Apply** or **OK** button.

After you have successfully solved for a specified *LSLOAD* (GUI field **Apply at LS**) and eventually *LSLOCK* (GUI field **Lock at LS**) value, you cannot modify that loading's settings during subsequent steps of the analysis. Similarly, you cannot delete loadings that you have already partially or completely solved.

You can select more than one pretension section at a time in order to specify identical loadings on them. Before you completely solve a given loading, any combination of pretension sections is valid. The following limitations apply:

- After you have completely solved one or more loadings, ANSYS allows multiple selection of *only* those pretension sections having

  -- the same number of defined loadings, *and*
  -- the identical loading number from the most recent completely solved loading.
- A multiple selection meeting the necessary criteria retains the settings that are identical for all selected pretension sections and leaves all other fields blank.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pretnsn Sectn
Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Section Lds
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Section
Main Menu>Solution>Define Loads>Apply>Structural>Pretnsn Sectn
Main Menu>Solution>Define Loads>Delete>All Load Data>All Section Lds
Main Menu>Solution>Define Loads>Delete>Structural>Section

SLPPLOT, *Prefer*, *VALUE*

Displays line loops smaller than a specified size (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

Prefer
Preference for line loop display. If *Prefer* = FACTOR, the command displays all line loops whose radius is smaller than the radius of the model (taken between the two most distant keypoints) times *VALUE*. This is the default preference. If *Prefer* = RADIUS, the command displays all line loops whose radius is smaller than that specified by *VALUE*.

VALUE
Numeric argument for *Prefer*.

Notes

Use this command to locate and display disproportionately small line loops when repairing the geometry of models imported from CAD files. Line loops matching the criteria specified in *Prefer* and *VALUE* both display in a different color and include their IDs. This command is available only for models imported through the Default IGES option.

Menu Paths

Main Menu>Preprocessor>Modeling>Simplify>Detect/Display>Small Loops

---

SLSPLOT, *Prefer*, *VALUE*

Displays line segments smaller than a specified size (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM <> FL PP ED

Prefer
Preference for line segment display. If *Prefer* = FACTOR, the command displays all line segments whose length is smaller than the average length of lines within the model times *VALUE*. This is the default preference. If *Prefer* = LENGTH, the command displays all line lengths smaller than that specified by *VALUE*.

VALUE
Numeric argument for *Prefer*.

Notes

Use this command to locate and display disproportionately small line segments when repairing the geometry of models imported from CAD files. Line lengths matching the criteria specified in *Prefer* and *VALUE* both display in a different color and include their IDs. This command is available only for models imported through the Default IGES option.

Menu Paths

Main Menu>Preprocessor>Modeling>Simplify>Detect/Display>Small Lines

---

SMALL, *IR*, *IA*, *IB*, *IC*, *Name*, --, --, *FACTA*, *FACTB*, *FACTC*

Finds the smallest of three variables.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA, *IB*, *IC*
Reference numbers of the three variables to be operated on. If only two, leave *IC* blank. If only one, leave *IB* blank also.

Name
Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --
Unused fields.

FACTA, *FACTB*, *FACTC*
Scaling factors (positive or negative) applied to the corresponding variables (defaults to 1.0).

Notes

Finds the smallest of three variables according to the operation:

IR = smallest of (*FACTA* x *IA*, *FACTB* x *IB*, *FACTC* x *IC*)

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Find Minimum

SMAX, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*

Forms an element table item from the maximum of two other items.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

LabR
Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

Lab1
First labeled result item in operation.

Lab2
Second labeled result item in operation (may be blank).

FACT1
Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

FACT2
Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

Notes

Forms a labeled result item (see **ETABLE** command) for the selected elements by comparing two existing labeled result items according to the operation:

LabR = (*FACT1* x *Lab1*) cmx (*FACT2* x *Lab2*)

where "cmx" means "compare and save maximum." If absolute values are requested [**SABS**,1], the absolute values of *Lab1* and *Lab2* are used.

Menu Paths

Main Menu>General Postproc>Element Table>Find Maximum

/SMBC, *Mode*

Controls the display of solid model boundary condition symbols and labels.

DATABASE: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Mode

CENT
: Solid model boundary condition symbols and labels appear at the centroid of the solid model entity (default).

TESS
: Solid model boundary condition symbols and labels appear inside each constituent element of the tesselation.

Notes

Mode = CENT is designed to reduce the clutter of boundary condition symbols in solid model plots. For example, if you have assigned normal pressure loads to an area, you may choose to display the pressures as arrows with the **/PSF** command using **/PSF**,PRES,NORM,2. When *Mode* = CENT, the pressure arrow is displayed at the centroid of the area. When *Mode* = TESS, a pressure arrow is displayed at the centroid of each polygon of the area's tesselation.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

SMBODY

Specifies "Body loads on the solid model" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Body Loads

SMCONS

Specifies "Constraints on the solid model" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>DOF Constraints

SMFOR

Specifies "Forces on the solid model" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Forces

SMIN, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*

Forms an element table item from the minimum of two other items.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

LabR
Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

Lab1
First labeled result item in operation.

Lab2
Second labeled result item in operation (may be blank).

FACT1
Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

FACT2
Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

Notes

Forms a labeled result item (see **ETABLE** command) for the selected elements by comparing two existing labeled result items according to the operation:

LabR = (*FACT1* x *Lab1*) cmn (*FACT2* x *Lab2*)

where "cmn" means "compare and save minimum." If absolute values are requested [**SABS**,1], the absolute values of *Lab1* and *Lab2* are used.

Menu Paths

Main Menu>General Postproc>Element Table>Find Minimum

SMOOTH, *Vect1*, *Vect2*, *DATAP*, *FITPT*, *Vect3*, *Vect4*, *DISP*

Allows smoothing of noisy data and provides a graphical representation of the data.

POST26: Special Purpose

MP ME ST <> <> PR EM <> FL PP ED

Vect1
Name of the first vector that contains the noisy data set (i.e., independent variable). You must create and fill this vector before issuing **SMOOTH**.

Vect2
Name of the second vector that contains the dependent set of data. Must be the same length as the first vector. You must create and fill this vector before issuing **SMOOTH**.

DATAP
Number of data points to be fitted, starting from the beginning of the vector. If left blank, the entire vector will be fitted. The maximum number of data points is 100,000 (or greater, depending on the memory of the computer).

FITPT
Order of the fitting curve that will be used as a smooth representation of the data. This number should be less than or equal to the number of the data points. Default (blank) is one-half the number of data points. Maximum number of smoothed data fitting order is the number of data points up to 50. Depending on this number, the smoothed curve will be one of the following:

1
Curve is the absolute average of all of the data points.

2
Curve is the least square average of all of the data points.

3 or more
Curve is a polynomial of the order (n-1), where n is the number of data fitting order points.

Vect3
Name of the vector that contains the smoothed data of the independent variable. This vector should have a length equal to or greater than the number of smoothed data points. In batch (command) mode, you must create this vector before issuing the **SMOOTH** command. In interactive mode, the GUI automatically creates this vector (if it does not exist). If you do not specify a vector name, the GUI will name the vector smth_ind.

Vect4
Name of the vector that contains the smoothed data of the dependent variable. This vector must be the same length as *Vect3*. In batch (command) mode, you must create this vector before issuing the **SMOOTH** command. In interactive mode, the GUI automatically creates this vector (if it does not exist). If you do not specify a vector name, the GUI will name the vector smth_dep.

DISP
Specifies how you want to display data. No default; you must specify an option.

1
Unsmoothed data only

2
Smoothed data only

3
Both smoothed and unsmoothed data

Notes

You can control the attributes of the graph using standard ANSYS controls (**/GRID**, **/GTHK**, **/COLOR**, etc.). If working interactively, these controls appear in this dialog box for convenience, as well as in their standard dialog boxes. You must always create *Vect1* and *Vect2* (using ***DIM**) and fill these vectors before smoothing the data. If you're working interactively, ANSYS automatically creates *Vect3* and *Vect4*, but if you're working in batch (command) mode, you must create *Vect3* and *Vect4* (using ***DIM**) before issuing **SMOOTH**. *Vect3* and *Vect4* are then filled automatically by ANSYS. In addition, ANSYS creates an additional TABLE type array that contains the smoothed array and the unsmoothed data to allow for plotting later with ***VPLOT**. Column 1 in this table corresponds to *Vect1*, column 2 to *Vect2*, and column 3 to *Vect4*. This array is named *Vect3*_SMOOTH, up to a limit of 32 characters. For example, if the array name is X1, the table name is X1_SMOOTH.

This command is also valid in PREP7 and SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Smooth Data
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Smooth Data
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Smooth Data
Main Menu>Solution>Loading Options>Smooth Data
Main Menu>TimeHist Postpro>Smooth Data

SMRTSIZE, *SIZLVL*, *FAC*, *EXPND*, *TRANS*, *ANGL*, *ANGH*, *GRATIO*, *SMHLC*, *SMANC*, *MXITR*, *SPRX*

Specifies meshing parameters for automatic (smart) element sizing.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

SIZLVL

Overall element size level for meshing. The level value controls the fineness of the mesh. (Any input in this field causes remaining arguments to be ignored.) Valid inputs are:

n

Activate SmartSizing and set the size level to *n*. Must be an integer value from 1 (fine mesh) to 10 (coarse mesh). Remaining arguments are ignored, and argument values are set as shown in SMRTSIZE - Argument Values for h-elements and p-elements.

STAT

List current **SMRTSIZE** settings.

DEFA

Set all **SMRTSIZE** settings to default values (as shown in SMRTSIZE - Argument Values for h-elements and p-elements for size level 6).

OFF

Deactivate SmartSizing. Current settings of **DESIZE** will be used. To reactivate SmartSizing, issue **SMRTSIZE**,*n*.

FAC

Scaling factor applied to the computed default mesh sizing. Defaults to 1 for h-elements (size level 6), which is medium. Values from 0.2 to 5.0 are allowed.

EXPND

Mesh expansion (or contraction) factor. (This factor is the same as **MOPT**,EXPND,*Value*.) *EXPND* is used to size internal elements in an area based on the size of the elements on the area's boundaries. For example, issuing **SMRTSIZE**,,,2 before meshing an area will allow a mesh with elements that are approximately twice as large in the interior of an area as they are on the boundary. If *EXPND* is less than 1, a mesh with smaller elements on the interior of the area will be allowed. *EXPND* should be greater than 0.5 but less than 4. *EXPND* defaults to 1 for h-elements (size level 6), which does not allow expansion or contraction of internal element sizes (except when using **AESIZE** element sizing). (If *EXPND* is set to zero, the default value of 1 will be used.) The actual size of the internal elements will also depend on the *TRANS* option or upon **AESIZE** or **ESIZE** sizing, if used.

TRANS

Mesh transition factor. (This factor is the same as **MOPT**,TRANS,*Value*.) *TRANS* is used to control how rapidly elements are permitted to change in size from the boundary to the interior of an area. *TRANS* defaults to 2.0 for h-elements (size level 6), which permits elements to approximately double in size from one element to the next as they approach the interior of the area. (If *TRANS* is set to zero, the default value will be used.) *TRANS* must be greater than 1 and, for best results, should be less than 4. The actual size of the internal elements will also depend on the *EXPND* option or upon **AESIZE** or **ESIZE** sizing, if used.

ANGL

Maximum spanned angle per lower-order element for curved lines. Defaults to 22.5 degrees per element (size level 6). This angle limit may be exceeded if the mesher encounters a small feature (hole, fillet, etc.). (This value is NOT the same as that set by **DESIZE**,,,,*ANGL*.) This option does not apply to p-element meshes.

ANGH
Maximum spanned angle per higher-order element for curved lines. Defaults to 30 degrees per element (size level 6). This angle limit may be exceeded if the mesher encounters a small feature (hole, fillet, etc.). (This value is NOT the same as that set by **DESIZE**,,,,,*ANGH*.)

GRATIO
Allowable growth ratio used for proximity checking. Defaults to 1.5 for h-elements (size level 6). Values from 1.2 to 5.0 are allowed; however, values from 1.5 to 2.0 are recommended.

SMHLC
Small hole coarsening key, can be ON (default for size level 6) or OFF. If ON, this feature suppresses curvature refinement that would result in very small element edges (i.e., refinement around small features).

SMANC
Small angle coarsening key, can be ON (default for all levels) or OFF. If ON, this feature restricts proximity refinement in areas where it is ill-advised (that is, in tight corners on areas, especially those that approach 0 degrees).

MXITR
Maximum number of sizing iterations (defaults to 4 for all levels).

SPRX
Surface proximity refinement key, can be off (*SPRX* = 0, *which is the default for all levels*) or on via two different values (*SPRX* = 1 or *SPRX* = 2). If *SPRX* = 1, surface proximity refinement is performed and any shell elements that need to be modified are modified. If *SPRX*=2, surface proximity refinement is performed but no shell elements are altered.

Command Default

SmartSizing is off and **DESIZE** is used for automatic element sizing.

Notes

If a valid level number (1 (fine) to 10 (coarse)) is input on *SIZLVL*, inputs for remaining arguments are ignored, and the argument values are set as shown in SMRTSIZE - Argument Values for h-elements and p-elements.

Note — The settings are different for h-elements and p-elements (indicated by the "h" and "p" headings in the table). The first column contains *SIZLV* data, ranging from 10 (coarse) to 1 (fine); 6 is the default.

SMRTSIZE - Argument Values for h-elements and p-elements

	FAC		EXPND		TRANS		ANGL		ANGH		GRATIO		SMHLC		SMANC		MXITR		SPRX	
	h	p	h	p	h	p	h	p	h	p	h	p	h	p	h	p	h	p	h	p
10	5.0	5.0	2.0	2.5	2.0*	2.0	45.0	-	45.0*	30.0	2.0	2.0	on	on	on	on	4*	4	off	off
9	3.0	4.0	1.75	2.35	2.0*	2.0	36.0	-	45.0*	30.0	1.9	1.9	on	on	on	on	4*	4	off	off
8	1.875	3.0	1.5	2.25	2.0*	2.0	30.0	-	45.0*	30.0	1.8	1.8	on	on	on	on	4*	4	off	off
7	1.5	2.5	1.0	1.7	2.0*	2.0	26.0	-	36.0*	30.0	1.7	1.7	on	on	on	on	4*	4	off	off
6*	1.0*	1.875	1.0*	1.5	2.0*	2.0	22.5	-	30.0*	30.0	1.5*	1.7	on	on	on	on	4*	4	off	off
5	0.65	1.5	1.0*	1.25	2.0*	2.0	18.0	-	27.0	30.0	1.5	1.6	on	on	on	on	4*	4	off	off
4	0.4	1.0	1.0*	1.0	2.0*	2.0	15.0	-	22.0	30.0	1.5	1.5	off	on	on	on	4*	4	off	off
3	0.3	0.8	1.0*	1.0	2.0*	2.0	12.0	-	18.0	22.0	1.5	1.5	off	off	on	on	4*	4	off	off

	FAC		EXPND		TRANS		ANGL		ANGH		GRATIO		SMHLC		SMANC		MXITR		SPRX	
	h	p	h	p	h	p	h	p	h	p	h	p	h	p	h	p	h	p	h	p
2	0.25	0.6	1.0*	1.0	2.0*	2.0	10.0	-	15.0	18.0	1.5	1.5	off	off	on	on	4*	4	off	off
1	0.2	0.4	1.0*	1.0	2.0*	2.0	7.5	-	15.0	15.0	1.4	1.4	off	off	on	on	4*	4	off	off

*Default

Where appropriate, SmartSizing will start with **AESIZE** settings. Elsewhere, it will start with any defined **ESIZE**,SIZE setting. It will locally override **AESIZE** or **ESIZE** for proximity and curvature. SmartSizing ignores any **ESIZE**,,NDIV setting.

LESIZE line division and spacing specifications will be honored by SmartSizing, unless you give permission for SmartSizing to override them (for proximity or curvature) by setting KYNDIV to 1. Lines not having an **LESIZE** specification are meshed as well as they can be.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Adv Opts
Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Basic
Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Status

SMSURF

Specifies "Surface loads on the solid model" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Surface Loads

SMULT, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*

Forms an element table item by multiplying two other items.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

LabR

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

Lab1
First labeled result item in operation.

Lab2
Second labeled result item in operation (may be blank).

FACT1
Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

FACT2
Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

Notes

Forms a labeled result item (see **ETABLE** command) for the selected elements by multiplying two existing labeled result items according to the operation:

LabR = (*FACT1* x *Lab1*) x (*FACT2* x *Lab2*)

May also be used to scale results for a single labeled result item. If absolute values are requested [**SABS**,1], the absolute values of *Lab1* and *Lab2* are used.

Menu Paths

Main Menu>General Postproc>Element Table>Multiply

SOLCONTROL, *Key1*, *Key2*, *Key3*, *Vtol*

Specifies whether to use optimized nonlinear solution defaults and some enhanced internal solution algorithms.

SOLUTION: Analysis Options

MP ME ST <> <> PR EM <> <> PP ED

Key1
Optimized defaults activation key:

ON or 1
Activates optimized defaults for a set of commands applicable to nonlinear solutions. This is the default. The majority of solution command defaults are listed under the Notes section below. See also the description of individual solution commands for default values.

OFF or 0
Restores defaults to pre-ANSYS 5.4 values (see the Default States table below). Internal solution algorithms work as for pre-ANSYS 5.4.

Key2
Check contact state key. This key is operable only when the optimized defaults are active (*Key1* = ON) and a contact or nonlinear status element is present in the model. When check contact state is active, ANSYS will base the time step size on the specifications of KEYOPT(7) for all contact elements. KEYOPT(7) for contact elements can be used to ensure that the time step interval accounts for changes in the contact status. Also, when *Key2* = ON, ANSYS ensures the time step is small enough to account for changes in nonlinear element status (applies to LINK10, COMBIN7, COMBIN14, COMBIN39, and COMBIN40 elements). Valid arguments for the key are:

ON or 1
Activate time step predictions based on specifications of element KEYOPT(7) or the nonlinear status of the element (applies to LINK10, COMBIN7, COMBIN14, COMBIN39, and COMBIN40 elements).

OFF or 0
Time step predictions not based on contact status or nonlinear element status (default).

Key3
Pressure load stiffness key. In general, use the default setting. Use a non-default setting only if you encounter convergence difficulties. Pressure load stiffness is automatically included when using eigenvalue buckling analyses (**ANTYPE**,BUCKLE), equivalent to *Key3* = INCP. For all other types of analyses, valid arguments for *Key3* are:

NOPL
Do not include pressure load stiffness for any elements.

no entry (default)
Include pressure load stiffness for elements SURF153, SURF154, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187 BEAM188, BEAM189, SHELL208, and SHELL209.

Do not include pressure load stiffness for elements PLANE2, PLANE42, SOLID45, SOLID46, SOLID64, SOLID65, PLANE82, VISCO88, VISCO89, SOLID92, SOLID95, and SOLID191.

INCP
Include pressure load stiffness for elements PLANE2, PLANE42, SOLID45, SOLID46, SOLID64, SOLID65, PLANE82, VISCO88, VISCO89, SOLID92, SOLID95, SURF153, SURF154, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187 BEAM188, BEAM189, SOLID191, SHELL208, and SHELL209.

Vtol
Tolerance for volumetric compatibility check in the 18x plane and solid elements (PLANE182, PLANE183, SOLID185, SOLID186, SOLID187) when mixed u-P formulation is employed. The default value is 1.0×10^{-5}. *Vtol* can range from 0.0 to 1.0, but values between 1.0×10^{-5} and 1.0×10^{-2} are recommended. For more details, see the *ANSYS, Inc. Theory Reference*.

Notes

The **SOLCONTROL** command is designed to provide reliable and efficient default solution settings for single-field full structural nonlinear or full transient analysis, or single-field thermal analysis. (It is not applicable for reduced transient analysis.) The optimized default settings and advanced internal solution algorithms can be used to solve the majority of structural/thermal, nonlinear/transient problems with the least amount of user intervention. The **SOLCONTROL** command is ON by default. In most cases, to successfully solve a problem the user only needs to:

- Choose **NLGEOM**,ON for large displacement/strain analysis.
- Choose **NROPT**,UNSYM to access unsymmetric pressure load stiffness, material stiffness, friction behavior, etc.
- Provide the initial step size through the **NSUBST** or **DELTIM** command.

To achieve this, the **SOLCONTROL** command chooses better default settings for a number of commands within ANSYS and uses optimized internal solution algorithms.

Commands that ANSYS issues are written to files (such as log and load step files) as **COMMAND**,-1 (for example, **KBC**,-1).

Note — The state of the **SOLCONTROL** command is not written by the **CDWRITE** or **LSWRITE** commands (so that the **CDWRITE** file does not rigidly define an individual solution command). Also note that switching **SOLCONTROL** ON and OFF between load steps is not recommended.

The **SOLCONTROL** command also serves as a "reset" command; when you issue **SOLCONTROL**, all the control commands set earlier in the interactive or input session are reset to their original default values.

Text database files (**.CDB** files created by **CDWRITE**) and load step files (**.S01**, **.S02**, **.S*nn***, etc. files created by **LSWRITE**) should be handled with care when **SOLCONTROL**,ON (default). Files of these types sometimes contain control commands that you did not issue intentionally. These extra commands can overwrite the default settings specified by **SOLCONTROL**. To avoid overwriting the **SOLCONTROL** settings when you are using **.CDB** files, use the following procedure:

1. Read the **.CDB** files into ANSYS.
2. Enter the solution processor [**/SOLU**].
3. Issue **SOLCONTROL**,ON.
4. Issue the desired control commands to overwrite the **SOLCONTROL** defaults as needed.

To use **.S*nn*** files properly, you should preview and edit them. Delete the unwanted solution commands before you execute the **LSSOLVE** command.

The following table lists the nonlinear solution parameters and algorithm differences when the command is active and inactive.

SOLCONTROL - Default States Table

Command	Argument	SOLCONTROL ON	SOLCONTROL OFF
General Options			
NSUBST	NSBSTP	1 load step if contact elements TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, CONTA174, CONTA175 are not present; if these elements are present, 1 or 20 substeps, depending on the physics of the program	Previously specified value. If no specified value, defaults to 1.
	NSBMX	Determined by ANSYS	Previously specified value. If no specified value, defaults to NSBSTP.
	NSBMN	Determined by ANSYS	Previously specified value. If no specified value, defaults to 1.
	Carry	Determined by ANSYS	OFF
DELTIM	DTIME	1 time span of the load step if contact elements TARGE169, TARGE170, CONTA171, CONTA172, CONTA173, CONTA174, CONTA175 are not present; if these elements are present, 1 or 1/20 of the time span of the load step, depending on the physics of the program	Previously specified value, if any.
	DTMIN	Determined by ANSYS	Previously specified value. If no specified value, defaults to DTIME

	DTMAX	Determined by ANSYS	Previously specified value. If no specified value, defaults to time span of load step.
	Carry	Determined by ANSYS	OFF
Command	**Argument**	**SOLCONTROL ON**	**SOLCONTROL OFF**
KBC	KEY	0 (ramped) for static nonlinear, structural and thermal (steady state) analyses, as well as for transient analyses when **TIMINT**,OFF. 1 (stepped) for transient structural and thermal analyses when **TIMINT**,ON. (**TIMINT**,ON is the default for transient analyses.)	0 for all types of transient or nonlinear analysis
AUTOTS	Key	Chosen by program	OFF
EQSLV		Uses sparse solver. If PCG solver is chosen, sets multiplier to 2.0 for Newton-Raphson iteration.	Uses sparse solver. If PCG solver is chosen, sets multiplier to 1.0 for Newton-Raphson iteration.
CDWRITE and **LSWRITE**		Does not write default values for most of the relevant solution control commands or options listed in this table.	Write all the default values for solution control commands.
MONITOR		Active	Not available
Nonlinear Options			
CNVTOL	TOLER	Force or moment convergence tolerance = 0.5% Displacement tolerance = 5%	Force or moment convergence tolerance = 0.1% Displacement tolerance not checked.
	MINREF	0.01 for force or moment; for heat flow and others the same as **SOLCONTROL**,OFF	For force or moment, 1.0 for heat flow, 1.0E-6 otherwise, 0
NEQIT	NEQIT	Between 15 and 26, depending on the physics of the problem.	25
ARCLEN		A more aggressive scheme to open-up time step is used. A more stable **ARCLEN** algorithm is used.	Use **ARCLEN** as in Release 5.3.
PRED	Sskey	On, unless ROTX, ROTY, and ROTZ are present, or SOLID65 is present.	OFF
LNSRCH	Key	Automatically turned ON when contact elements present.	OFF
CUTCONTROL	PLSLIMIT	15%	5%
	NPOINT	13	20
OPNCONTROL	TEMP	.01	Not Available
	NUMSTEP	3	Not Available
SSTIF	Key	ON for geometrically nonlinear analysis (**NLGEOM**, ON).	OFF
NROPT	ADPTKY	OFF, except: when frictional contact exists; when elements CONTAC12 or CONTAC52 are present; or when plasticity exists and one of the elements PIPE20, BEAM23, BEAM24, or PIPE60 is present.	Automatically toggled on and off depending on whether plasticity or frictional contact exists or not.

Command	Argument	SOLCONTROL ON	SOLCONTROL OFF
TINTP	THETA	1.0	.5
	TOL	0.0	.2
Element Options			
CONTAC12, CONTAC52		Time prediction independent of KEYOPT(7) (default) except when requested.	Time prediction depending on KEYOPT(7).
CONTAC12, CONTAC52		Adaptive descent ON when friction is present.	Same
PIPE20, BEAM23, BEAM24, PIPE60		Adaptive descent ON when plasticity is present.	Same
BEAM4, SHELL63, SHELL143		Consistent tangent KEYOPT(2) = 1 when **NLGEOM**,ON.	KEYOPT(2) = 0
Algorithm Behavior			
Deformed element shape (Jacobi) check used as criteria for early bisection		Active	Not available
Euler backward theta (for first order equations)		1.0 for thermal analysis.	0.0
Log file		Does not write default values for any of the relevant commands or options listed in this table.	Write all the default values for solution controls commands.
Moment reference values		Automatically corrected when values become too small by using a reaction force times an element characteristic length.	When zero **CONVTOL**, MINREF value is used.
Automatic time step scheme		Check on nonconvergent patterns. Time step is opened up less aggressively; the increase factor (used in calculating the degree in which the time step is opened) = 1.5 (in most cases). The calculation also takes into account the physics of the problem.	Check on nonconvergent patterns not implemented. Time step is opened up more aggressively; the increase factor = 2.0 (in most cases). No physics dependency involved.
Reset of all solution control defaults in one command.		**SOLCONTROL**,ON or OFF	Not available
Nonlinear convergence criterion		When force norm is smaller than 1, the calculated force value is still used as the REF value. If the calculated force value is approaching machine zero, the MINREF value is used as REF.	When force norm is smaller than 1, the MINREF value is used as the REF value.
Warning message printed when negative diagonal in matrix is discovered.		Simplified message, not printed in some cases.	Detailed message printed for each iteration.
Stop button		Available in GUI. **Jobname.ABT** file can also be used.	Use **Jobname.ABT** file to control normal abort.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Mixed U-P Toler

Main Menu>Preprocessor>Loads>Load Step Opts>Solution Ctrl
Main Menu>Solution>Load Step Opts>Nonlinear>Mixed U-P Toler
Main Menu>Solution>Load Step Opts>Solution Ctrl

SOLU, *NVAR*, *Item*, *Comp*, *Name*

Specifies solution summary data per substep to be stored.

POST26: Set Up

MP ME ST DY <> PR EM <> <> PP ED

NVAR
Arbitrary reference number assigned to this variable (2 to *NV* [**NUMVAR**]).

Item
Label identifying the item. Valid item labels are shown in the table below. Some items may also require a component label.

Comp
Component of the item (if required). Valid component labels are shown in the table below. None are currently required.

Name
Thirty-two character name identifying the item on printouts and displays. Defaults to an eight character label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Notes

See also the **PRITER** command of POST1 to display some of these items directly. Valid for a static or full transient analysis. All other analyses have zeros for the data. Valid item and component labels for solution summary values are:

Valid Item Labels

Item	Description
ALLF	Total arc-length load factor (ratio of the load at equilibrium to the total applied load)
ALDLF	Arc-length load factor increment (change in ALLF)
ARCL	Normalized arc-length radius
CNVG	Convergence indicator.
CRPRAT	Maximum creep ratio.
CSCV	Current segment convergence value.
CUCV	Current convergence value.
DICV	Displacement convergence value.
DSPRM	Descent parameter.
DTIME	Time step size.
EQIT	Number of equilibrium iterations.
FFCV	Fluid flow convergence value.
FOCV	Force convergence value.
HFCV	Heat flow convergence value.
NCMIT	Cumulative number of iterations.

Valid Item Labels

Item	Description
NCMLS	Cumulative number of load steps
NCMSS	Cumulative number of substeps
MFCV	Magnetic flux convergence value.
MOCV	Moment convergence value.
MXDVL	Maximum degree of freedom value.
PRCV	Pressure convergence value.
PSINC	Maximum plastic strain increment.
RESFRQ	Response frequency for 2nd order systems.
RESEIG	Response eigenvalue for 1st order systems.
ROCV	Rotation convergence value.
SMCV	Scalar magnetic potential convergence value.
TECV	Temperature convergence value.
VECV	Velocity convergence value.
VOCV	Voltage convergence value.
VMCV	Vector magnetic potential convergence value.

Menu Paths

Main Menu>TimeHist Postpro>Define Variables
Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

/SOLU

Enters the solution processor.

SESSION: Processor Entry
SOLUTION: Analysis Options

MP ME ST DY <> PR EM <> FL PP ED

Notes

This command is valid only at the Begin Level.

Menu Paths

Main Menu>Solution

SOLUOPT

Specifies "Solution options" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>General

SOLVE

Starts a solution.

SOLUTION: Analysis Options

MP ME ST DY <> PR EM EH FL <> ED

Notes

Starts the solution of one load step of a solution sequence based on the current analysis type and option settings.

Menu Paths

Main Menu>DesignXplorer VT>Solution>Solve
Main Menu>Drop Test>Solve
Main Menu>Solution>Run FLOTRAN
Main Menu>Solution>Solve
Main Menu>Solution>Solve>Current LS

SORT

Specifies "Sort settings" as the subsequent status topic.

POST1: Status

MP ME ST DY <> PR EM <> FL <> ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Sort Module

SOURCE, *X*, *Y*, *Z*

Defines a default location for undefined nodes or keypoints.

PREP7: Keypoints
PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

X, *Y*, *Z*
Global Cartesian coordinates for source nodes or keypoints (defaults to the origin).

Command Default

Global Cartesian origin.

Notes

Defines a global Cartesian location for undefined nodes or keypoints moved during intersection calculations [**MOVE** or **KMOVE**].

Menu Paths

This command cannot be accessed from a menu.

SPACE, *NODE*

Defines a space node for radiation using the Radiation Matrix method.

AUX12: Radiation Substructures

MP ME ST <> <> PR <> <> <> PP ED

NODE
Node defined to be the space node.

Command Default

No space node (no radiation to space).

Notes

A space node is required in an open system to account for radiation losses.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Other Settings

SPARM, *Porti*, *Portj*

Calculates scattering (S) parameters between ports of a network system.

POST1: Magnetics Calculations

MP <> <> <> <> <> <> EH <> PP ED

Porti
Port number of the excited port with a excitation mode. (See the description of the **HFPORT** command.)

Portj
Port number of the output port. This could be used for a multiport system. All ports but *Porti* must be matched.

Notes

SPARM calculates scattering (S) parameters for multiport or single port (Porti = Portj) systems.

The **SPARM** command macro returns the following complex S parameters: SII, dBSII, SIIPMR, SJI, dBSJI, and SJIPMR, where "I" is the port number for the excited port, "J" is the output port number and PMR is the phase angle.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Port>S-Parameters

SPCNOD, *ENCL*, *NODE*

Defines a space node for radiation using the Radiosity method.

SOLUTION: Radiosity

MP ME <> <> <> PR <> <> <> PP ED

ENCL
Radiating surface enclosure number. Defaults to 1. If ENCL = STAT, the command lists all enclosure space nodes. If ENCL = DELE, the command deletes all enclosure space nodes.

NODE
Node defined to be the space node.

Notes

For open systems, an enclosure may radiate to a space node (*NODE*).

Open systems may be characterized by one or more enclosures (*ENCL*). Each enclosure may radiate to a different space node (*NODE*).

For a space node that is not part of the finite element model, specify the temperature using the **D** command. For the first load step, the space node temperature ramps from the uniform temperature specified by the **TUNIF** command to the temperature specified by the **D** command. For subsequent load steps, it ramps from the previous value of the space node temperature. For intermediate load steps, use the **SPCNOD**,DELETE command and specify the space node temperature again to ramp from the uniform temperature.

For a space node that is part of the finite element model, the temperature is that calculated during the finite element solution.

Menu Paths

Main Menu>Preprocessor>Radiation Opts>Solution Opt
Main Menu>Radiation Opt>Radiosity Meth>Solution Opt
Main Menu>Solution>Radiation Opts>Solution Opt

SPCTEMP, *ENCL*, *TEMP*

Defines a free-space ambient temperature for radiation using the Radiosity method.

SOLUTION: Radiosity

MP ME <> <> <> PR <> <> <> PP ED

ENCL
Radiating surface enclosure number. Defaults to 1. If *ENCL* = STAT, the command lists all enclosure space temperatures. If *ENCL* = DELE, the command deletes all enclosure space temperatures.

TEMP
Temperature of free-space in the reference temperature system. The temperature will be offset by the value specified in the **TOFFST** command for internal calculations.

Notes

For open systems, an enclosure may radiate to the free-space ambient temperature (*TEMP*).

Open systems may be characterized by one or more enclosures (*ENCL*). Each enclosure may radiate to a different free-space ambient temperature (*TEMP*).

For the first load step, the space temperature ramps from the uniform temperature specified by the **TUNIF** command to the temperature specified by the **SPCTEMP** command. For subsequent load steps, it ramps from the previous value of the space temperature. For intermediate load steps, use the **SPCTEMP**,DELETE command and specify the space temperature again to ramp from the uniform temperature.

Menu Paths

Main Menu>Preprocessor>Radiation Opts>Solution Opt
Main Menu>Radiation Opt>Radiosity Meth>Solution Opt
Main Menu>Solution>Radiation Opts>Solution Opt

SPEC

Specifies "Miscellaneous specifications" as the subsequent status topic.

POST1: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Output Options

SPH4, *XCENTER*, *YCENTER*, *RAD1*, *RAD2*

Creates a spherical volume anywhere on the working plane.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

XCENTER, *YCENTER*

Working plane X and Y coordinates of the center of the sphere.

RAD1, *RAD2*

Inner and outer radii (either order) of the sphere. A value of zero or blank for either *RAD1* or *RAD2* defines a solid sphere.

Notes

Defines either a solid or hollow spherical volume anywhere on the working plane. The sphere must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) A sphere of 360° will be defined with two areas, each consisting of a hemisphere. See the **SPHERE** and **SPH5** commands for other ways to create spheres.

When working with a model imported from an IGES file (DEFAULT import option), you can create only solid spheres. If you enter a value for both *RAD1* and *RAD2* the command is ignored.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Primitives>Solid Sphere
Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>Hollow Sphere
Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>Solid Sphere
Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Sphere>Solid Sphere

---

SPH5, *XEDGE1*, *YEDGE1*, *XEDGE2*, *YEDGE2*
Creates a spherical volume by diameter end points.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

XEDGE1, *YEDGE1*
Working plane X and Y coordinates of one edge of the sphere.

XEDGE2, *YEDGE2*
Working plane X and Y coordinates of the other edge of the sphere.

Notes

Defines a solid spherical volume anywhere on the working plane by specifying diameter end points. The sphere must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) A sphere of 360° will be defined with two areas, each consisting of a hemisphere. See the **SPHERE** and **SPH4** commands for other ways to create spheres.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>By End Points
Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Sphere>By End Points

---

SPHERE, *RAD1*, *RAD2*, *THETA1*, *THETA2*
Creates a spherical volume centered about the working plane origin.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

RAD1, *RAD2*
Inner and outer radii (either order) of the sphere. A value of zero or blank for either *RAD1* or *RAD2* defines a solid sphere.

THETA1, *THETA2*
Starting and ending angles (either order) of the sphere. Used for creating a spherical sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0.0° and the ending angle defaults to 360.0°. See the *ANSYS Modeling and Meshing Guide* for an illustration.

Notes

Defines either a solid or hollow sphere or spherical sector centered about the working plane origin. The sphere must have a spatial volume greater than zero. (i.e., this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) Inaccuracies can develop when the size of the object you create is much smaller than the relative coordinate system values (ratios near to or greater than 1000). If you require an exceptionally small sphere, create a larger object, and scale it down to the appropriate size.

For a solid sphere of 360°, you define it with two areas, each consisting of a hemisphere. See the **SPH4** and **SPH5** commands for the other ways to create spheres.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>By Dimensions
Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Create>Volume>Sphere>By Dimensions

---

SPLINE, *P1*, *P2*, *P3*, *P4*, *P5*, *P6*, *XV1*, *YV1*, *ZV1*, *XV6*, *YV6*, *ZV6*

Generates a segmented spline through a series of keypoints.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

P1, *P2*, *P3*, *P4*, *P5*, *P6*
Keypoints through which the spline is fit. At least two must be defined. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

The following fields are used only if specified end slopes on the line are desired, otherwise zero curvature end slopes will be automatically calculated to produce a line which is "straight" in the active coordinate system. To specify end slopes, use the following fields to define a "slope vector" (one for each end of the line, if desired) that has its tail at the origin and its head at the point *XVn*,*YVn*, *ZVn* in the active coordinate system [**CSYS**]. The corresponding end slope of the line will then be parallel to this "slope vector."

XV1, *YV1*, *ZV1*
Location (in the active coordinate system) of the head of the "slope vector" corresponding to the slope at the *P1* end of the spline. The tail of the vector is at the origin of the coordinate system.

XV6, *YV6*, *ZV6*
Location of the head of the "slope vector" corresponding to the slope at the *P6* (or the last keypoint if fewer than six specified) end of the spline.

Notes

The output from this command is a series of connected lines (one line between each pair of keypoints) that together form a spline. Note that solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Splines>Segmented Spline
Main Menu>Preprocessor>Modeling>Create>Lines>Splines>With Options>Segmented Spline

---

SPLOT, *NA1*, *NA2*, *NINC*, *MESH*

Displays the selected areas and a faceted view of their underlying surfaces

PREP7: Areas

MP ME ST DY <> PR EM <> FL PP ED

NA1
Starting area for display of areas and underlying surfaces. If *NA1* = ALL (default), *NA2* and *NINC* are ignored and all selected areas are displayed (**ASEL** command).

NA2
Last area to be displayed.

NINC
Numeric value setting steps between NA1 and NA2 for display. Default value is (1).

MESH
Specifies a rectangular mesh density used to display the underlying surface (default 4, i.e. 4 x 4).

Notes

This command is valid in any processor. The plot output displays the external and internal trim curves and underlying surface. You cannot obtain a faceted view of your surface areas when you are using the **/EXPAND** command to create larger graphics displays.

Use **APLOT** for trimmed surface display.

Menu Paths

This command cannot be accessed from a menu.

SPOINT, *NODE*, *X*, *Y*, *Z*

Defines a point for moment summations.

POST1: Special Purpose

MP ME ST <> <> PR <> <> <> PP ED

NODE
Node number of the desired point. If zero, use *X*,*Y*,*Z* to describe point.

X, *Y*, *Z*
Global Cartesian coordinates of the desired summation point. Used if *NODE* is 0. Defaults to (0,0,0).

Command Default

No point is defined by default; you must either specify a node or coordinates.

Notes

Defines a point (any point other than the origin) about which the tabular moment summations are computed [**NFORCE**, **FSUM**]. If force summations are desired in other than the global Cartesian directions, a node number must be specified on the *NODE* field, *and* the desired coordinate system must be activated with **RSYS**.

Menu Paths

Main Menu>General Postproc>Nodal Calcs>Summation Pt>At Node
Main Menu>General Postproc>Nodal Calcs>Summation Pt>At XYZ Loc

---

SPOPT, *Sptype*, *NMODE*, *Elcalc*

Selects the spectrum type and other spectrum options.

SOLUTION: Spectrum Options

MP ME ST <> <> PR <> <> <> PP ED

Sptype
Spectrum type:

SPRS
Single point excitation response spectrum (default). See also the **SVTYP** command.

MPRS
Multiple point excitation response spectrum.

DDAM
Dynamic design analysis method.

PSD
Power spectral density.

NMODE
Use the first *NMODE* modes from the modal analysis. Defaults to all extracted modes, as specified by the **MODOPT** and **BUCOPT** commands. *NMODE* cannot be larger than 1000.

Elcalc
Element calculation key (for *Sptype* = PSD only):

NO
Do not include stress responses in the calculations (default).

YES
Include stress responses in the calculations.

Notes

Valid only for a spectrum analysis (**ANTYPE**,SPECTR). This operation must be preceded by a modal solution (**ANTYPE**,MODAL) with the appropriate files available. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Product Restrictions

Only *Sptype* = SPRS is allowed in ANSYS Professional.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

SPREAD, *VALUE*

Turns on a dashed tolerance curve for the subsequent curve plots.

POST26: Display

MP ME ST DY <> PR EM <> FL PP ED

VALUE
Amount of tolerance. For example, 0.1 is ± 10%.

Command Default

No tolerance curve.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph

SPSCAN, *FREQ*, *LOCAL*, *PHIB*, *PHIE*, *PHIINC*, *THETAB*, *THETAE*, *THETAINC*, *FILEOPT*

Performs a harmonic analysis of a cellar unit over a range of angles and extracts the S-parameter.

SOLUTION: Analysis Options

MP <> <> <> <> <> <> EH <> PP ED

FREQ
Working frequency in Hz.

LOCAL
Local coordinate system number (defaults to 0).

PHIB
Beginning angle from x-axis towards y-axis, ϕ, in degrees (defaults to 0).

PHIE
Ending angle from x-axis towards y-axis, ϕ, in degrees (defaults to 0).

PHIINC
Increment of ϕ in degrees (defaults to 0).

THETAB
Beginning angle from +z-axis towards -z-axis, θ, in degrees (defaults to 0).

THETAE
Ending angle from +z-axis towards -z-axis, θ, in degrees (defaults to 0).

THETAINC
Increment of θ in degrees (defaults to 0).

FILEOPT

0
Output S-parameter magnitude and phase angle to **Jobname.scan**.

1
Output S-parameter magnitude in dB and phase angle to **Jobname.scan**.

Notes

See the figure Figure 4.23: "Spherical Coordinates" in the *ANSYS Low-Frequency Electromagnetic Analysis Guide* for an illustration of the coordinate system.

Menu Paths

SPSWP, *FREQB*, *FREQE*, *FREQINC*, *SwpOpt*, *EFAcc*, *OutPut*, *FileOpt*

Computes S-parameters over a frequency range and writes them to a file.

SOLUTION: Analysis Options

MP <> <> <> VT <> <> EH <> PP ED

FREQB
Frequency (Hz) at the beginning of the *FREQB* to *FREQE* range. If *FREQE* is blank, the solution is done only at frequency *FREQB*.

FREQE
Frequency at end of this range.

FREQINC
Frequency increment. The number of solutions performed is [(*FREQE* - *FREQB*)/*FREQINC*] + 1. Solutions are always performed at *FREQB* and *FREQE*.

SwpOpt

0
Variational Technology (See **HROPT** command) (default).

1
Full method (See **HROPT** command).

Note — To use the Variational Technology method, you must have the separately licensed ANSYS Frequency Sweep VT (FS Module).

EFAcc
Electric field accuracy for fast exploration method. Defaults to 0.01.

OutPut

0
Create minimal size results file required to compute S-parameters for the last port excitation case (see Notes), for all frequencies. (default).

1
Create complete results file for the last port excitation case (see Notes), for all frequencies.

2
Create complete results file for the last port excitation case (see Notes), for the last frequency only, valid only for full method.

FileOpt

0
Output S-parameter magnitude and phase angle in Touchstone file.

1
Output S-parameter magnitude in dB and phase angle in Touchstone file.

2
Output S-parameter real and imaginary parts in Touchstone file.

Notes

This command computes S-parameters by sequencing through a series of harmonic solutions with different port excitations over the desired frequency range. To use this function, port flag boundary conditions must be previously set (see **SF**, **SFA** for exterior waveguide ports, or **BFA**, **BFL** or **BF** for interior waveguide or transmission line ports). Ports should be numbered sequentially from "1" with no gaps in the numbering. Resulting S-parameters are written to a file **jobname.s*n*p**, where *n* is the number of ports. The file is written in Touchstone format.

S-parameters require a sequence of solutions whereby for each solution, one port is "excited" and the other ports are "matched". Each solution represents one column of a S-parameter matrix (i.e., if Port 1 is excited for a three-port system, the resulting column represents the S11, S21 and S31 S-parameters). A full S-parameter matrix for an *n* port system at one frequency requires *n* solutions alternating "excited" and "matched" port boundary conditions at each port. The **SPSWP** command will solve a column of the S-parameter matrix for each port that has a defined excitation. If all ports have a defined excitation, then the full S-parameter matrix will be computed.

To prepare the ports for **SPSWP**, each port must be defined (flagged) as a port and it must have a port excitation defined. For waveguide ports, excitation is defined using the **HFPORT** command. For transmission line ports, excitation is defined using the **BFA**, **BFL**, or **BF** commands using the JS, H, or EF load options. Transmission line excitation must be defined on the port surface itself (nodes, lines, or areas).

Menu Paths

Main Menu>Solution>Solve>S-Par Sweep

SPTOPT

Specifies "Spectrum analysis options" as the subsequent status topic.

SOLUTION: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Show Status
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Show Status
Utility Menu>List>Status>Solution>Spectrum Options

SQRT, *IR*, *IA*, --, --, *Name*, --, --, *FACTA*

Forms the square root of a variable.

POST26: Operations

MP ME ST DY <> PR EM <> FL PP ED

IR
Arbitrary reference number assigned to the resulting variable (2 to *NV* [**NUMVAR**]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA
Reference number of the variable to be operated on.

--, --
Unused fields.

Name
Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --
Unused fields.

FACTA
Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

Notes

Forms the square root of a variable according to the operation:

$$IR = \sqrt{FACTA \times IA}$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Square Root

SRSS, *SIGNIF*, *Label*

Specifies the square root of sum of squares mode combination method.

SOLUTION: Spectrum Options

MP ME ST <> <> PR <> <> <> PP ED

SIGNIF
Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single point, multi-point, or DDAM response (**SPOPT**,SPRS, MPRS or DDAM), the significance level of a mode is defined as the mode coefficient of the mode, divided by the maximum mode coefficient of all modes. Any mode whose

significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT**,PSD.)

Label
Label identifying the combined mode solution output.

DISP
Displacement solution (default). Displacements, stresses, forces, etc., are available.

VELO
Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

ACEL
Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

Notes

This command is also valid for PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine
Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine

---

/SSCALE, *WN*, *SMULT*

Sets the contour multiplier for topographic displays.

GRAPHICS: Scaling

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

SMULT
Contour multiplier that factors in results based on the product of the multiplier and the result being plotted. Defaults to 0.0 (no topographic effects).

Command Default

No topographic contour effects.

Notes

Use this command to scale values to the geometry when the contours are shown elevated. For section displays [**/TYPE**], the elevation is performed perpendicular to the section face.

Nonzero contour multipliers factoring in large results (stresses or displacements) can produce very large distortion, causing images to disappear. To bring a distorted image back into view, reduce the contour multiplier value.

Portions of this command are not supported by PowerGraphics [**/GRAPHICS**,POWER].

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Contour Style

SSLN, *FACT*, *SIZE*

Selects and displays small lines in the model.

PREP7: Lines

MP ME ST DY <> PR EM <> FL PP ED

FACT

Factor used to determine small lines. *FACT* times the average line length in the model is used as the line length limit below which lines will be selected.

SIZE

Line length limit for line selection. Lines that have a length less than or equal to *SIZE* will be selected. Used only if *FACT* is blank.

Notes

SSLN invokes a predefined ANSYS macro for selecting small lines in a model. Lines that are smaller than or equal to the specified limit (*FACT* or *SIZE*) are selected and line numbers are displayed. This command macro is useful for detecting very small lines in a model that may cause problems (i.e., poorly shaped elements or a meshing failure) during meshing. All lines that are not "small" will be unselected and can be reselected with the **LSEL** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>Sel Small Lines

SSTIF, *Key*

Activates stress stiffness effects in a nonlinear analysis.

SOLUTION: Nonlinear Options

MP ME ST <> <> PR <> <> <> PP ED

Key

Stress stiffening key:

OFF

No stress stiffening is included (default unless **NLGEOM**,ON).

ON

Stress stiffening is included (default if **NLGEOM**,ON).

Command Default

SSTIF will be turned on if **NLGEOM**,ON; otherwise it will be turned off.

Notes

Activates stress stiffness effects in a nonlinear analysis (**ANTYPE**,STATIC or TRANS). (The **PSTRES** command also controls the generation of the stress stiffness matrix and therefore should not be used in conjunction with **SSTIF**.) If used in SOLUTION, this command is valid only within the first load step.

When **SOLCONTROL** and **NLGEOM** are ON, **SSTIF** defaults to ON. This normally forms all of the consistent tangent matrix. However, for some special nonlinear cases, this can lead to divergence caused by some elements which do not provide a complete consistent tangent (notably, elements outside the 18*x* family). In such a case, ANSYS recommends issuing an **SSTIF**,OFF command to achieve convergence. For the 18*x* family of elements, setting **SSTIF**,OFF when **NLGEOM** is ON has no effect (because stress stiffness effects are always included).

The default values given for this command assume **SOLCONTROL**,ON (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL**,ON and **SOLCONTROL**,OFF.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

SSUM

Calculates and prints the sum of element table items.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

Notes

Calculates and prints the tabular sum of each existing labeled result item [**ETABLE**] for the selected elements. If absolute values are requested [**SABS**,1], absolute values are used.

Menu Paths

Main Menu>General Postproc>Element Table>Sum of Each Item

STAOPT, *Method*

Specifies static analysis options.

SOLUTION: Analysis Options

MP ME ST <> VT PR <> <> <> PP ED

Method

Solution method for the static analysis:

DEFA
Regular ANSYS solve (default).

SX
Solve with Variational Technology.

Notes

Specifies the method of solution for a static analysis (**ANTYPE**,STATIC). If used in SOLUTION, this command is valid only within the first load step.

After a solution based on Variational Technology with the **SOLVE** command, the Variational Technology model is "frozen", i.e., you cannot add or delete input variables.

This command is also valid in PREP7.

Menu Paths

Main Menu>DesignXplorer VT>Solution>Solve

STAT

Displays the status of database settings.

DATABASE: Set Up
DISPLAY: Action
MP ME ST DY <> PR EM <> FL PP ED

Notes

In the DISPLAY program, **STAT** will show the current status of the program settings.

In the ANSYS program, **STAT** is a command generated by the GUI and will appear in the log file (**Jobname.LOG**) if status is requested for some items under **Utility Menu> List> Status**. Generally, **STAT** will be preceded by one of the commands listed below, which specifies the particular topic that status was requested for.

If entered directly into the program, the **STAT** command should be immediately preceded by the desired topic command listed below. In processors other than those listed below (e.g., AUX12), no topic command should proceed **STAT**.

This command is valid in any processor.

PREP7 topic commands (and their corresponding topics) are:

Topic Command	Topic
ETYPE	Element types
RCON	Real constants
MATER	Material properties
TBLE	Data table properties
PRIM	Solid model primitives
KEYPTS	Keypoints

Topic Command	Topic
LINE	Lines
AREAS	Areas
VOLUMES	Volumes
GEOMETRY	Solid model information
MESHING	Meshing
BOOL	Booleans
NODES	Nodes
ELEM	Elements
SELM	Superelements
PIPE	Pipe modeling
DIGIT	Node digitizing
COUPLE	Node coupling
CEQN	Constraint equations
REORDER	Model reordering

SOLUTION topic commands (and their corresponding topics) are:

Topic Command	Topic
ATYPE	Analysis types
MASTER	Master DOF
GAP	Reduced transient gap conditions
DEACT	Element birth and death (deactivation)
LSOPER	Load step operations
FECONS	Constraints on nodes
FEFOR	Forces on nodes
FESURF	Surface loads on elements
FEBODY	Body loads on elements
SMCONS	Constraints on the solid model
SMFOR	Forces on the solid model
SMSURF	Surface loads on the solid model
SMBODY	Body loads on the solid model
INRTIA	Inertial loads
GENOPT	General options
DYNOPT	Dynamic analysis options
NLOPT	Nonlinear analysis options
OUTOPT	Output options
BIOOPT	Biot-Savart options
SPTOPT	Spectrum analysis options
SOLUOPT	Solution options
FLOTRAN	FLOTRAN data settings

POST1 topic commands (and their corresponding topics) are:

Topic Command	Topic
DEFINE	Data definition settings
SORT	Sort settings
PRINT	Print settings
DISPLAY	Display settings
CALC	Calculation settings
PATH	Path data settings
LCCALC	Load case settings
DATADEF	Directly defined data status
FATIGUE	Fatigue data status
POINT	Point flow tracing settings
SPEC	Miscellaneous specifications

POST26 topic commands (and their corresponding topics) are:

Topic Command	Topic
DEFINE	Data definition settings
OPERATE	Operation data
PRINT	Print settings
PLOTTING	Plotting settings

Menu Paths

Main Menu>Prob Design>Prob Database>Status
Main Menu>Solution>Solve>Current LS

/STATUS, *Lab*

Lists the status of items for the run.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Lab
Items to list status for:

ALL
List all below (default).

TITLE
List only titles, **Jobname**, and revision number.

UNITS
List only units.

MEM
List only memory data statistics.

DB
List only database statistics

CONFIG
List only configuration parameters.

GLOBAL
Provides a global status summary.

SOLU
Provides a solution status summary.

PROD
Provides a product summary.

Notes

Displays various items active for the run (such as the ANSYS revision number, **Jobname**, titles, units, configuration parameters, database statistics, etc.).

This command is valid in any processor.

Menu Paths

Utility Menu>List>Status>Global Status

STEF, *VALUE*

Specifies Stefan-Boltzmann radiation constant.

AUX12: Radiation Substructures
SOLUTION: Radiosity

MP ME ST <> <> PR <> <> <> PP ED

VALUE
Stefan-Boltzmann constant (defaults to 0.119E-10 Btu/hr/in^2/ °R^4).

Command Default

0.119E-10 Btu/hr/in^2/ °R^4.

Notes

You can use this command in the general preprocessor (PREP7) and in the Solution processor to specify the Stefan-Boltzmann constant for a FLOTRAN analysis using radiation surface boundary conditions.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Displacement>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Displacement>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Displacement>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Displacement>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Forces>Body Forces>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Turbulence>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Turbulence>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Turbulence>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Velocity>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Velocity>On Keypoints
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Velocity>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Velocity>On Nodes
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Volume Fract>Bound Loads>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/CFD>Volume Fract>Bound Loads>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Nodes
Main Menu>Preprocessor>Radiation Opts>Solution Opt
Main Menu>Radiation Opt>Matrix Method>Other Settings
Main Menu>Radiation Opt>Radiosity Meth>Solution Opt
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Displacement>On Areas
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Displacement>On Keypoints
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Displacement>On Lines
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Displacement>On Nodes
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Forces>Body Forces>On Nodes
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Turbulence>On Areas
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Turbulence>On Lines
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Turbulence>On Nodes
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Velocity>On Areas
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Velocity>On Keypoints
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Velocity>On Lines
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Velocity>On Nodes
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Volume Fract>Bound Loads>On Elements
Main Menu>Solution>Define Loads>Apply>Fluid/CFD>Volume Fract>Bound Loads>On Lines
Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Areas
Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Elements
Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Lines
Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Nodes
Main Menu>Solution>Radiation Opts>Solution Opt

/STITLE, *NLINE*, *Title*

Defines subtitles.

DATABASE: Set Up

MP ME ST DY <> PR EM <> FL PP ED

NLINE
Subtitle line number (1 to 4). Defaults to 1.

Title
Input up to 70 alphanumeric characters. Parameter substitution may be forced within the title by enclosing the parameter name or parametric expression within percent (%) signs. If *Title* is blank, this subtitle is deleted.

Notes

Subtitles (4 maximum) are displayed in the output along with the main title [**/TITLE**]. Subtitles do not appear in GUI windows or in ANSYS plot displays. The first subtitle is also written to various ANSYS files along with the main title. Previous subtitles may be overwritten or deleted. Issue **/STATUS** to display titles.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

STORE, *Lab*, *NPTS*

Stores data in the database for the defined variables.

POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Lab

Valid labels:

MERGE
: Merge data from results file for the time points in memory with the existing data using current specifications (default).

NEW
: Store a new set of data, replacing any previously stored data with current result file specifications and deleting any previously-calculated (OPER) variables. Variables defined using the **ANSOL** command are also deleted.

APPEN
: Append data from results file to the existing data.

ALLOC
: Allocate (and zero) space for *NPTS* data points.

PSD
: Create a new set of frequency points for PSD calculations (replacing any previously stored data and erasing any previously calculated data).

NPTS

The number of time points (or frequency points) for storage (used only with *Lab* = ALLOC or PSD). The value may be input when using POST26 with data supplied from other than a results file. This value is automatically determined from the results file data with the NEW, APPEN, and MERGE options. For the PSD option, *NPTS* determines the resolution of the frequency vector (valid numbers are between 1 and 10, defaults to 5).

Command Default

Merge newly-defined variables with previously stored variables for the time points stored in memory using the current specifications. If **STORE** is preceded by **TIMERANGE** or **NSTORE**, the default is **STORE**,NEW.

Notes

This command stores data from the results file in the database for the defined variables [**NSOL**, **ESOL**, **SOLU**] per specification [**FORCE**, **LAYERP26**, **SHELL**]. See the *ANSYS Basic Analysis Guide* for more information.

The **STORE**,PSD command will create a new frequency vector (variable 1) for response PSD calculations [**RPSD**]. This command should first be issued before defining variables [**NSOL**, **ESOL**, **RFORCE**] for which response PSD's are to be calculated.

Menu Paths

Main Menu>TimeHist Postpro>Store Data

SUBOPT, *SUBSIZ*, *NPAD*, *NPERBK*, *NUMSSI*, *NSHIFT*, *Strmck*, *JCGITR*

Specifies options for subspace iteration eigenvalue extraction.

SOLUTION: Nonlinear Options
SOLUTION: Dynamic Options

MP ME ST <> <> PR <> <> <> PP ED

SUBSIZ
Subspace working size. Defaults to *NMODE* + 4 (where *NMODE* is input on the **MODOPT** or **BUCOPT** command). Minimum is 8. Maximum is *NMODE* +*NPAD*. The larger the value, the smaller the number of iterations (but more time per iteration).

NPAD
Number of extra vectors used in the iterations. Defaults to 4. The total number of vectors used is *NMODE* +*NPAD*.

NPERBK
Number of modes per memory block. If 0 (or blank), perform data management in-memory for all modes (no disk I/O). If greater than zero, use some disk I/O (slower for decreasing *NPERBK* values, but may be needed for large problems). The minimum nonzero value is the number of degrees of freedom per node for the model.

NUMSSI
Maximum number of subspace iterations (defaults to 100). Fewer iterations will be done if convergence occurs before the 100th iteration. Convergence occurs whenever the normalized change in the eigenvalue calculations between successive iterations for the first *NMODE* eigenvalues is less than 1.0E-5.

NSHIFT
Minimum number of subspace iterations completed before a shift is performed. The default is 5 and the minimum is 2. Use *FREQB* on the **MODOPT** command or *SHIFT* on the **BUCOPT** command to define the initial shift point.

Strmck
Sturm sequence check key:

ALL
Perform check at all shift points as well as at the end point (default).

PART
Perform check only at all shift points.

NONE
Do not perform Sturm sequence check.

JCGITR
Number of Jacobi iterations used per subspace iteration (used only with the JCG and PCG options on the **EQSLV** command). Defaults to the number of degrees of freedom divided by the maximum wave front for the model. The minimum is 5.

Command Default

As described for the option defaults above.

Notes

Defines options for subspace iteration eigenvalue extraction (**MODOPT**,SUBSP or **BUCOPT**,SUBSP). Default values should be satisfactory for most solutions. See the *ANSYS, Inc. Theory Reference* for option details. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

SUBSET, *Lstep*, *SBSTEP*, *FACT*, *KIMG*, *TIME*, *ANGLE*, *NSET*

Reads results for the selected portions of the model.

POST1: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Lstep
Load step number of the data set to be read (defaults to 1):

N
Read load step *N*.

FIRST
Read the first data set (*SBSTEP* and *TIME* are ignored).

LAST
Read the last data set (*SBSTEP* and *TIME* are ignored).

NEXT
Read the next data set (*SBSTEP* and *TIME* are ignored). If at the last data set, the first data set will be read as the next.

NEAR
Read the data set nearest to *TIME* (*SBSTEP* is ignored). If *TIME* is blank, read the first data set.

LIST
Scan the results file and list a summary of each load step. (*FACT*, *KIMG*, *TIME* and *ANGLE* are ignored.)

SBSTEP
Substep number (within *Lstep*). For the buckling (**ANTYPE**,BUCKLE) analysis or the modal (**ANTYPE**,MODAL) analysis, the substep corresponds to the mode number. Defaults to last substep of load step (except for **ANTYPE**,BUCKLE or MODAL). If *Lstep* = LIST, *SBSTEP* = 0 or 1 lists the basic step information, whereas *SBSTEP* = 2 also lists the load step title, and labels imaginary data sets if they exist.

FACT
Scale factor applied to data read from the file. If zero (or blank), a value of 1.0 is used. Harmonic velocities or accelerations may be calculated from the displacement results from a modal (**ANTYPE**,MODAL) or harmonic response (**ANTYPE**,HARMIC) analyses. If *FACT* = VELO, the harmonic velocities (v) are calculated from the displacements (d) at a particular frequency (f) according to the relationship $v = 2 \pi fd$. Similarly, if *FACT* = ACEL, the harmonic accelerations (a) are calculated as $a = (2 \pi f)^2 d$.

KIMG
Used only with results from complex analyses:

0
Store real part of complex solution

1
Store imaginary part.

TIME
Time-point identifying the data set to be read. For the harmonic responses analyses, time corresponds to the frequency. For the buckling analysis, time corresponds to the load factor. Used only in the following cases: If *Lstep* is NEAR, read the data set nearest to *TIME*. If both *Lstep* and *SBSTEP* are zero (or blank), read data set at time = *TIME*. If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets. Solution items not written to the results file [**OUTRES**] for either data set will result in a null item after data set interpolation. If *TIME* is beyond the last time point on the file, use the last time point.

ANGLE
Circumferential location (0.0 to 360°). Defines the circumferential location for the harmonic calculations used when reading from the results file. The harmonic factor (based on the circumferential angle) is applied to the harmonic elements (PLANE25, PLANE75, PLANE78, FLUID81, PLANE83, and SHELL61) of the load case. See the *ANSYS, Inc. Theory Reference* for details. Note that factored values of applied constraints and loads will overwrite any values existing in the database.

NSET
Data set number of the data set to be read. If a positive value for *NSET* is entered, *Lstep*, *SBSTEP*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by ***SET**,LIST.

Notes

Reads a data set from the results file into the database for the selected portions of the model only. Data that has not been specified for retrieval from the results file by the **INRES** command will be listed as having a zero value. Each time that the **SUBSET** command is issued, the data currently in the database will be overwritten with a new set of data. Various operations may also be performed during the read operation. The database must have the model geometry available (or used the **RESUME** command before the **SUBSET** command to restore the geometry from **File.DB**).

Menu Paths

Main Menu>General Postproc>Read Results>By Load Step

Main Menu>General Postproc>Read Results>By Set Number
Main Menu>General Postproc>Read Results>By Time/Freq

SUCALC, *RSetName*, *lab1*, *Oper*, *lab2*, *fact1*, *fact2*, *const*

Create new result data by operating on two existing result data sets on a given surface.

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

RSetName
Eight character name for new result data.

lab1
First result data upon which to operate.

Oper
Mathematical operation to perform.

ADD
(*lab1* + *lab2* + *const*)

SUB
(*lab1* - *lab2* + *const*)

MULT
(*lab1* * *lab2* + *const*)

DIV
(*lab1* / *lab2* + *const*)

EXP
(*lab1* ^ *fact1* + *lab2* ^ *fact2* + *const*)

COS
(cos (*lab1*) + *const*)

SIN
(sin (*lab1*) + *const*)

ACOS
(acos (*lab1*) + *const*)

ASIN
(asin (*lab1*) + *const*)

ATAN
(atan (*lab1*) + *const*)

ATAN2
(atan2 (*lab1*) + *const*)

LOG
(log (*lab1*) + *const*)

ABS
(abs (*lab1*) + *const*)

ZERO
(0 + *const*)

lab2
Second result data upon which to operate.

fact1
First scaling factor (for EXP option only).

fact2
Second scaling factor (for EXP option only).

const
Constant added to the values in the resulting data.

Menu Paths

Main Menu>General Postproc>Surface Operations>Math Operations>Absolute
Main Menu>General Postproc>Surface Operations>Math Operations>Add
Main Menu>General Postproc>Surface Operations>Math Operations>ArcCosine
Main Menu>General Postproc>Surface Operations>Math Operations>ArcSine
Main Menu>General Postproc>Surface Operations>Math Operations>ArcTangent
Main Menu>General Postproc>Surface Operations>Math Operations>ArcTangent2
Main Menu>General Postproc>Surface Operations>Math Operations>Cosine
Main Menu>General Postproc>Surface Operations>Math Operations>Divide
Main Menu>General Postproc>Surface Operations>Math Operations>Exponentiate
Main Menu>General Postproc>Surface Operations>Math Operations>Initialize
Main Menu>General Postproc>Surface Operations>Math Operations>Multiply
Main Menu>General Postproc>Surface Operations>Math Operations>Natural Log
Main Menu>General Postproc>Surface Operations>Math Operations>Sine
Main Menu>General Postproc>Surface Operations>Math Operations>Subtract

---

SUCR, *SurfName*, *SurfType*, *nRefine*, *Radius*

Create a surface.

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

SurfName
Eight character surface name.

SurfType
Surface type.

CPLANE
Surface is defined by cutting plane in window one (controlled by the working plane (/CPLANE,1) and NOT view settings (/CPLANE,0)).

SPHERE
Surface is defined by a spherical surface centered about the working plane origin.

nRefine
Refinement level.

For SurfType = CPLANE
nRefine is the refinement level of the surface "mesh" and is an integer between 0 and 3 (default = 0). See Notes below.

For SurfType = SPHERE
nRefine is the number of divisions along a 90° arc (minimum = 9, maximum = 90), default is 9.

Radius
Radius of sphere (for *SurfType* = SPHERE).

Notes

This is an action command. This command will store the following quantities for the defined surface:

GCX, GCY, GCZ are global Cartesian coordinates at each point on the surface.
NORMX, NORMY, NORMZ are components of the unit normal at each point on the surface.
DA is the contributory area of each point.

For *SurfType* = CPLANE, *nRefine* refers to the number of points that define the surface.

If *SurfType* = CPLANE and *nRefine* = 0, the points reside at the section cuts through the element by the cutting plane.

Increasing *nRefine* to 1 will subdivide each surface facet into 4 subfacets, thus increasing the number of points at which result can be interpolated.

Menu Paths

Main Menu>General Postproc>Surface Operations>Create Surface>On Cuttng Plane
Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node
Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>By Dimensions

---

SUDEL, *SurfName*

Delete geometry information as well as any mapped results for specified surface.

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

SurfName
Eight character surface name.

SurfName = ALL will delete all surface geometry and result infromation.

Menu Paths

Main Menu>General Postproc>Surface Operations>Delete Surfaces

---

SUEVAL, *Parm*, *lab1*, *Oper*

Perform operations on a mapped item and store result in a scalar parameter.

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

Parm
APDL parameter name.

lab1
Eight character set name for the first set used in calculation.

Oper
Operation to perform:

SUM
Sum of *lab1* result values.

INTG
Integral of *lab1* over surface.

AVG
Area-weighted average of a result item [Σ(*lab1**DA) / Σ(DA)]

Notes

The result of this operation is a scalar APDL parameter value. If multiple surfaces are selected when this command is issued, then the operation is carried out on each surface individually and the parameter reperesents the cul-mulative value of the operation on all selected surfaces.

Menu Paths

Main Menu>General Postproc>Surface Operations>Math Operations>Average Result
Main Menu>General Postproc>Surface Operations>Math Operations>Integrate Results
Main Menu>General Postproc>Surface Operations>Math Operations>Sum of Results

---

SUGET, *SurfName*, *RSetName*, *Parm*, *Geom*

Moves surface geometry and mapped results to an array parameter.

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

SurfName
Eight character surface name.

RSetName
Eight character result name.

Parm
APDL array parameter name (up to 32 characters).

Geom
Switch controlling how data is written.

ON/1/YES
Writes geometry data and interpolated results information to the parameter.

OFF/0/NO
Writes only interpolated results information to the parameter. (Default)

Notes

For *Geom* = OFF/0/NO, only results information is written to this parameter.

For *Geom* = ON/1/YES, both geometry data and results information are written to this parameter. Geometry data includes 7 data items: (GCX, GCY, GCZ, NORMX, NORMY, NORMZ and DA). Results information is then written to the 8th column of the parameter. SetNames of GCX, GCY, GCZ, NORMX, NORMY, NORMZ and DA are predefined and computed when **SUCR** is issued.

Menu Paths

Main Menu>General Postproc>Surface Operations>Results to Array

---

SUMAP, *RSetName*, *Item*, *Comp*

Map results onto selected surface(s).

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

RSetName
Eight character name for the result being mapped.

Item
Label identifying the item.

Valid item labels are defined in **PLNSOL**. Some items also require a component label.

If *Item* = CLEAR, the specified result set will be deleted from all selected surfaces

Comp
Component label of item (if required).

Notes

Results are mapped in current **RSYS**. This command interpolates and stores results data on to each of the selected surfaces. **SUMAP**, ALL, CLEAR will delete all results sets from all selected surfaces.

Menu Paths

Main Menu>General Postproc>Surface Operations>Clear Results

SUMTYPE, *Label*

Sets the type of summation to be used in the following load case operations.

POST1: Results

MP ME ST DY <> PR EM <> FL PP ED

Label
Summation type

COMP
Combine element component stresses only. Stresses such as average nodal stresses, principal stresses, equivalent stresses, and stress intensities are derived from the combined element component stresses. Default.

PRIN
Combine principal stress, equivalent stress, and stress intensity directly as stored on the results file. Component stresses are not available with this option.

Notes

Issue **SUMTYPE**,PRIN when you want to have a load case operation (**LCOPER**) act on the principal / equivalent stresses instead of the component stresses.

Menu Paths

Main Menu>General Postproc>Load Case>Calc Options>Stress Options

---

SUPL, *SurfName*, *RSetName*, *KWIRE*

Plot result data on all selected surfaces or on a specified surface.

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

SurfName
Eight character surface name. *ALL* will plot all selected surfaces.

RSetName
Eight character result name.

KWIRE
Plot in context of model.

0
Plot results without the outline of selected elements.

1
Plot results with the outline of selected elements.

Notes

If *RSetName* is left blank, then the surface geometry will be plotted. If the Setname portion of the argument is a vector prefix (i.e. if result sets of name SetNameX, SetNameY and SetNameZ exist), ANSYS will plot these vectors

on the surface as arrows. For example, **SUPL**, ALL, NORM will plot the surface normals as vectors on all selected surfaces, since NORMX NORMY and NORMZ are pre-defined geometry items.

Menu Paths

Main Menu>General Postproc>Surface Operations>Plot Results
Main Menu>General Postproc>Surface Operations>Plot Vectors

SUPR, *SurfName*, *RSetName*

Print global status, geometry information and/or result information.

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

SurfName
Eight character surface name. If *SurfName* = ALL, repeat printout for all selected surfaces.

RSetName
Eight character result set name.

Notes

When no arguments are specified, **SUPR** generates a global status summary of all defined surfaces. If only *SurfName* is specified, the geometry information for that surface is printed. If both *SurfName* and *RSetName* are specified, the value of the results set at each point, in addition to the geometry information, is printed.

Menu Paths

Main Menu>General Postproc>Surface Operations>Print Results
Main Menu>General Postproc>Surface Operations>Status>Global

SURESU, --, *Fname*, *Fext*, *Fdir*

Read a set of surface definitions and result items from a file and make them the current set.

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

--
Unused field

Fname
Eight character name.

Fext
Extension name.

Fdir
Optional path specification.

Notes

Reading (and therefore resuming) surface and result definitions from a file overwritea any existing surface definitions.

Reading surfaces back into the postprocessor (**/POST1**) does *not* insure that the surfaces (and their results) are appropriate for the model currently residing in **/POST1**.

Menu Paths

Main Menu>General Postproc>Surface Operations>Resume Surfaces

---

SUSAVE, *Lab*, *Fname*, *Fext*, *Fdir*

Saves surface definitions to a file.

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

Lab
Eight-character surface name.

If *Lab* = ALL (default), then all surfaces are saved to the file.

If *Lab* = S, only currently selected surfaces are saved to the file.

Fname
File name and directory path (248 character maximum, including directory). If you do not specify a directory path, the default is your working directory and you can use all 248 characters for the file name. The file name defaults to the jobname.

Fext
File name extension (eight-character maximum). The extension defaults to "surf".

Fdir
Optional path specification.

Notes

The **SUSAVE** command saves surface definitions (geometry information)--and any result items mapped onto the surfaces--to a file.

Issuing the **SUSAVE** command has no effect on the database. The database remains unchanged.

Subsequent executions of the **SUSAVE** command overwrite previous data in the file.

To read the contents of the file created via the **SUSAVE** command, issue the **SURESU** command.

Menu Paths

Main Menu>General Postproc>Surface Operations>Save Surfaces

SUSEL, *Type*, *Name1*, *Name2*, *Name3*, *Name4*, *Name5*, *Name6*, *Name7*, *Name8*

Selects a subset of surfaces

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of select:

S
Selects a new set (default).

R
Reselects a set from the current set.

A
Additionally selects a set and extends the current set.

U
Unselects a set from the current set.

ALL
Also selects all surfaces.

NONE
Unselects all surfaces.

Name1, *Name2*, *Name3*, *Name4*, *Name5*, *Name6*, *Name7*, *Name8*
Eight character surface names

Notes

The selected set of surfaces is used in the following operations: **SUMAP**, **SUDEL**, **SUCALC**, **SUEVAL**, and **SUVECT**.

Menu Paths

Main Menu>General Postproc>Surface Operations>Select Surfaces

---

SUVECT, *RSetName*, *lab1*, *Oper*, *lab2*, *Offset*

Create new result data by operating on two existing result vectors on a given surface.

POST1: Surface Operations

MP ME ST DY <> PR EM <> FL PP ED

RSetName
Eight character name of the result data output. There will be one or three *RSetName* values depending on the operation specified in *Oper*.

lab1
Eight character name of the mapped data that forms vector 1. Specified sets must exist on all selected surfaces for this operation to take place. The names NORM and GC will be reserved for normals and for global (x, y, z).

Oper

DOT
: Computes dot product between *lab1* and *lab2* vectors. The result is a scalar parameter (*RSetName*) and each value within the set can be modified (incremented) via *Offset*.

CROSS
: Computes cross product between *lab1* and *lab2* vectors. Each X, Y, Z value in the result can be modified (incremented) via *Offset*.

SMULT
: Scales (lab1x, lab1y, lab1z) vector by scalar *lab2*. Each X,Y,Z value in the result can be modified (incremented) via *Offset*.

lab2
: Eight character name of the mapped data that forms vector 2. Sets with names Lab2X, Lab2Y, and Lab2Z must exist on all selected surfaces for operation to take place. For *Oper* = SMULT a scalar value or another predefined scalar item (e.g., DA) can be supplied.

Offset
: An offset value to be applied to the resultant *RSetName*. One value is specified for *Oper* = DOT, and three values are specified for *Oper* = SMULT.

Menu Paths

Main Menu>General Postproc>Surface Operations>Math Operations>Cross Product
Main Menu>General Postproc>Surface Operations>Math Operations>Dot Product
Main Menu>General Postproc>Surface Operations>Math Operations>Scale Result

---

SV, *DAMP*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*, *SV8*, *SV9*
Defines spectrum values to be associated with frequency points.

SOLUTION: Spectrum Options

MP ME ST <> <> PR <> <> <> PP ED

DAMP
: Damping ratio for this response spectrum curve. If the same as a previously defined curve, the SV values are added to the previous curve. Up to four different curves may be defined, each with a different damping ratio. Damping values must be input in ascending order.

SV1, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*, *SV8*, *SV9*
: Spectrum values corresponding to the frequency points [**FREQ**]. Values are interpreted as defined with the **SVTYP** command. Log-log interpolation is used between curves. SV values should not be zero. Values required outside the frequency range use the extreme input values.

Notes

Defines the spectrum values to be associated with the previously defined frequency points [**FREQ**]. Applies only to the single-point response spectrum. Damping has no effect on the frequency solution. Damping values are used only to identify SV curves for the mode combinations calculation. Only the curve with the lowest damping value is used in the initial mode coefficient calculation. Use **STAT** command to list current spectrum curve values.

Repeat **SV** command for additional SV points (20 maximum per *DAMP* curve). SV values are added to the *DAMP* curve after the last nonzero SV value.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Spectr Values
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Spectr Values

---

SVTYP, *KSV*, *FACT*

Defines the type of single-point response spectrum.

SOLUTION: Spectrum Options

MP ME ST <> <> PR <> <> <> PP ED

KSV

Response spectrum type:

0
: Seismic velocity response spectrum loading (SV values interpreted as velocities with units of length/time).

1
: Force response spectrum loading (SV values interpreted as force amplitude multipliers).

2
: Seismic acceleration response spectrum loading (SV values interpreted as accelerations with units of length/time^2).

3
: Seismic displacement response spectrum loading (SV values interpreted as displacements with units of length).

4
: PSD loading (SV values interpreted as acceleration^2/(cycles/time), such as (in/sec^2)^2/Hz (not g^2/Hz)). (Not recommended)

FACT

Scale factor applied to spectrum values (defaults to 1.0). Values are scaled when the solution is initiated [**SOLVE**]. Database values remain the same.

Command Default

Seismic velocity response spectrum.

Notes

Defines the type of single-point response spectrum [**SPOPT**]. The seismic excitation direction is defined with the **SED** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Settings
Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Settings

/SX

Enters the Variational Technology preprocessor.

SESSION: Processor Entry
VARIATIONAL TECHNOLOGY: DesignXplorer VT
MP ME ST DY <> PR EM <> FL PP ED

Notes

Enters the Variational Technology processor.

Menu Paths

Main Menu>DesignXplorer VT>Solution>Solve

SXCLR, *Type*

Clears the Variational Technology database.

VARIATIONAL TECHNOLOGY: DesignXplorer VT
<> <> <> <> VT <> <> <> <> <> <>

Type
Specifies the part of the Variational Technology database to be cleared. Valid labels are:

ALL --
Clears the entire Variational Technology database. Both the preprocessing part (settings for input variables, result parameters, etc.) and results part (results for the derivatives) are cleared.

RSLT --
Clear only the results part of the Variational Technology database. This is necessary if you want to modify the Variational Technology model (for example, adding a new input variable and so on) after a solution has already been performed.

Notes

Clear the database of the Variational Technology module. The settings are reset to their default values and the memory is cleared. Remember that the result files containing the results of Variational Technology (see **SXRFIL** command) are never deleted unless you intentionally delete them. We recommend that you use this command before switching to a new solution, which, for example, includes different input variables or different element or node components for input variables. See the various Variational Technology commands for changes that are possible after the solution has been performed.

If the entire database is cleared with **SXCLR**,ALL then also all definitions for input variables, result parameters, solution method, result file storage and so on are either deleted and/or reset to their default values.

Menu Paths

Main Menu>DesignXplorer VT>Other>Clear Database

SXDISC, *Name*, *ElComp*

Defines an element component as a discrete input variable for the DesignXplorer VT.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

Name

Name for the input variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks are allowed in the string.

ElComp

The name of either of an element component or the number of a single element.

Command Default

None

Notes

Defines a component containing a group of elements as a discrete input variable of the DesignXplorer VT. In a discrete optimization the DesignXplorer VT will evaluate the dependency of the result parameter(s) (see the **SXRSLT** command) as a function of the discrete case of either including the group of elements in the finite-element model or excluding the group of elements from the model.

The *ElComp* field defines a group of elements that are considered as either being included or excluded from the finite-element model during a discrete optimization.

If a solution has already been performed with Variational Technology using **STAOPT**,SX and the **SOLVE** command, then the Variational Technology database is locked. In this case no input variables can be added to the model. You cannot change the specific definitions of an existing input variable. If you want to modify the Variational Technology model by adding a discrete variable, you first have to unlock the model with the **SXCLR** command.

Menu Paths

Main Menu>DesignXplorer VT>Setup>Discrete Variable

SXEVAL

Triggers evaluation of generated results based on input variables specified via the SXVMOD command.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

Notes

Used in conjunction with the **SXVMOD** command, **SXEVAL** evaluates the designated variables and loads the results into the ANSYS database. You can employ the ANSYS postprocessing tools to view the results, or perform an optimization or PDS loop.

Menu Paths

Main Menu> DesignXplorer VT> Postprocessing> Evaluate Results

---

SXFREQ, *Name*, *MIN*, *MAX*, *INC*, *RedOpt*

Defines the frequency as input variable for the FS Module or the DesignXplorer VT.

VARIATIONAL TECHNOLOGY: DesignXplorer VT
VARIATIONAL TECHNOLOGY: FS Module

<> <> <> <> VT <> <> <> <> <> <>

Name
Name of the variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks can be included in the string.

MIN
Minimum value of the input variable.

MAX
Maximum value of the input variable.

INC
Number of increments within the range of the input variable as defined by the *MIN* and *MAX* fields.

RedOpt
Reduction option that specifies what to do if the DesignXplorer VT needs to reduce the variation range from *MIN* to *MAX*. Valid labels are:

CONT
Continue with the calculation of the derivatives. This is the default.

STOP
Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

Command Default

There are no default values for the command fields.

Notes

Defines the frequency as an input variable. Variational Technology will evaluate a frequency sweep from the specified minimum to the maximum frequency values given in the *MIN* and *MAX* fields. The frequency sweep will include as many frequency increments as specified in the *INC* field.

If a solution has already been performed with Variational Technology using **STAOPT**,*SX* and the **SOLVE** command, then the Variational Technology database is locked. In this case no input variables can be added to the model. You cannot change the specific definitions of an existing input variable. If you want to modify the Variational Technology model by adding a discrete variable, you first have to unlock the model with the **SXCLR** command.

Menu Paths

Main Menu>DesignXplorer VT>Setup>Frequency

SXGEOM, *Name*

Defines a geometry parameter as a DesignXplorer VT input variable.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

Name

Name of an input variable. This is a string of up to 256 characters used for post-processing purposes. It can have blanks included in the string.

Notes

Defines a geometry parameter as an input variable of the DesignXplorer VT.

Menu Paths

This command cannot be accessed from a menu.

SXMETH, *SoluType*, *ApprType*, *ModeTrack*

Defines the solution options for the DesignXplorer VT.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

SoluType

Specifies the type of derivatives that need to be evaluated during the solution run of the DesignXplorer VT. Valid labels are:

INDP

Evaluates the derivatives assuming that the input variables are independent. Therefore, mixed derivatives will be neglected.

FULL

Evaluates all required derivatives including mixed derivatives, addressing the interactions between the input variables (if necessary). This option is the default.

ApprType
Specifies the type of approximation function that is used to build the DesignXplorer VT. Valid labels are:

PADE
Uses a series expansion based on Pade approximations using rational functions. This method is the more accurate, however it is limited to a single input variable per element.

TAYL
Uses a Taylor series expansion as the approximation function. This must be used when there is more than one input variable per element, i.e. Young's Modulus and thickness. When performing a normal nodes analysis, use the TAYL method for faster solution times.

AUTO
Use the PADE method whenever possible, TAYL method otherwise. This option is the default.

ModeTrack
Specifies whether or not mode tracking should be in a modal solution of the DesignXplorer VT. Valid labels are:

YES
Mode Tracking will be used in a modal solution. This option is the default.

NO
Mode Tracking will not be used in a modal solution.

Command Default

Defaults as outlined above.

Notes

Determines which derivatives of the result parameters with respect to the input variables shall be calculated during the DesignXplorer VT solution run. Also determines in which type of approximation function the derivatives are used.

If your Variational Technology analysis includes discrete variables defined with the SXDISC command, then ApprType=PADE will be used if you specify AUTO or PADE.

The default ApprType=AUTO is always the recommended and preferred approximation method. If using AUTO generates error messages, you must switch to the PADE or TAYL method.

If Variational Technology is not used in connection with a modal analysis, then the ModeTrack option is ignored.

Menu Paths

Main Menu>DesignXplorer VT>Solution>Solution Method

SXMP, *Name*, *MIN*, *MAX*, *RedOpt*, *Lab*, *MAT*, *ElComp*, *VarType*, *Order*

Defines a material property as an input variable for DesignXplorer VT.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

Name
Name for the input variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks can be included in the string.

MIN
Minimum value of the input variable.

MAX
Maximum value of the input variable.

RedOpt
Reduction option that specifies what to do if DesignXplorer VT needs to reduce the variation range from *MIN* to *MAX*. Valid labels are:

CONT
Continue with the calculation of the derivatives. This is the default.

STOP
Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

Lab
Valid material property label. This label must be defined for the material number *MAT* using the **MP** or **MPDATA** command prior to issuing this command. Applicable labels are:

EX
Elastic Moduli

NUXY
Minor Poisson's ratios

GXY
Shear moduli

DENS
Material density

MAT
Material identification number

ElComp
Name of an element component or number of a single individual element. If *ElComp* = ALL, then all elements are considered for the **SXMP** command.

VarType
Label indicating how the variability of temperature dependent material properties should be handled. This is particularly useful to describe how temperature dependent material properties are handled. At this release, only one label (the default) is valid:

FACT
The variability of the material property is described by a scale factor that is applied to the material property values. In the case of temperature dependent material properties, a scale factor will be applied

to all defined material property values. This means that the *MIN* field is the value of the minimum factor to be applied and the *MAX* field is the maximum factor to be applied. As such the values specified for *MIN* and *MAX* are unitless quantities. *VarType* = FACT is the default for temperature dependent material properties.

ADD
The variability of the material property is described by an add-on that is applied to the material property values. In the case of a temperature dependent material property, the defined property values are shifted all at the same time. This means that the *MIN* field is the minimum value that is added to the material property values and the *MAX* field contains the maximum value that is added to the material property values. Hence, the values specified for *MIN* and *MAX* have the same unit as the material property itself.

VAL
The variability of the material property is described by the variability of the material property value itself. This is only applicable if the material property is *not* temperature dependent. This means that the *MIN* field is the minimum value that the material property itself can take and the *MAX* field contains the maximum value which the material property values can take. This is the default if the material properties are *not* temperature dependent.

Order
Derivation order. By default, driven by the application.

Command Default

As described above.

Notes

Defines a material property of a single element or a set of elements as an input variable of the DesignXplorer VT. The DesignXplorer VT will evaluate the derivatives of the result parameter(s) (see **SXRSLT** command) with respect to the material property defined here. The derivatives are used to determine an approximation function for the result parameter(s) as a function of the material property in the range specified by the *MIN* and *MAX* fields.

The type of the material property is defined by the Lab field, which must not be left blank. The material property must exist or be specified using the **MP** or **MPDATA** command prior to issuing the **SXMP** command.

Using either the *MAT* field or the *ElComp* field a set of element is selected for which the material property is varied. The user must define one of them and leave the respective other one blank. All selected elements must be associated with the same material number. This is of course always given in the *MAT* field. If selected by the *ElComp* field, an error will be issued if an element is found that is associated with a material number that is different from the one associated with any other element in the element component.

If the material properties are temperature dependent, then the temperature dependency is defined by either several data points at different temperatures (**MP**) or by coefficients defining the temperature dependent polynomial (**MPDATA**). To describe a variation of the material property with one single parameter is only possible by using a scalar addition to the temperature dependency curve (*VarType* = ADD) or by using a scalar multiplication factor for the entire temperature dependency curve (*VarType* = FACT). Varying the material property value itself using *VarType* = VAL is not possible, if the material property is temperature dependent.

An error message is generated and the **SXMP** command is ignored when the current value is outside the variation range.

If the Order is not specified, it will be increased until the accuracy requirements of the **SXRSLT** command are satisfied. When you set a value for Order, no further adjustments for accuracy will be performed. Order should only be used when an input parameter has an influence with a limited order.

Menu Paths

Main Menu>DesignXplorer VT>Setup>Material Property

SXPOST

Launches the DesignXplorer VT postprocessing application.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

Notes

The postprocessing application provides a viewer for Variational Technology results files. You can only launch this application if the ANSYS GUI is active; that is, it cannot be launched from batch mode.

Menu Paths

Main Menu>DesignXplorer VT>Postprocessing>SolutionViewer

SXREAL, *Name*, *MIN*, *MAX*, *RedOpt*, *Lab*, *NSET*, *ElComp*, *VarType*, *Order*

Defines a real constant property as an input variable for the DesignXplorer VT.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

Name
Name of the input variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks are allowed in the string.

MIN
Minimum value of the input variable.

MAX
Maximum value of the input variable.

RedOpt
Reduction option that specifies what to do if the DesignXplorer VT needs to reduce the variation range from *MIN* to *MAX*. Valid labels are:

CONT
Continue with the calculation of the derivatives. This is the default.

STOP
Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

Lab

Valid real constant label. This label must refer to a valid property of the real constant set as defined in the **R** command. Applicable labels are:

TK

Shell thickness. This applies only to shell elements of type SHELL181, where the element thickness has been defined with the **R** command.

STIF

Spring stiffness. This applies only to combination elements of type COMBIN14, where the spring stiffness has been defined with the **R** command.

MASS

Lumped mass. This applies only to structural mass elements of type MASS21 with KEYOPT(3) = 2, where the lumped mass has been defined with the **R** command.

NSET

Real constant set identification number as defined by the **R** command.

ElComp

Name of an element component or number of a single individual element. If *ElComp* = ALL, then all elements are considered for the **SXREAL** command.

VarType

Label indicating how the variability of a spatially dependent element thickness should be handled. This label is ignored if the shell element thickness is constant for all elements selected by either the *NSET* or the *ElComp* field. Valid labels are:

FACT

The variability of the section property is described by a scale factor that is applied to the section property values. In case the section properties are different from element to element, a scale factor will be applied to all section property values. This means that the *MIN* field is the value of the minimum factor to be applied and the *MAX* field is the maximum factor to be applied. As such the values specified for *MIN* and *MAX* are unitless quantities. *VarType* = FACT is the default if the section property is different from element to element.

ADD

The variability of the section property is described by an add-on that is applied to the section property values. In case the section properties are different from element to element, then the section property values are shifted at the same time by this add-on. This means that the *MIN* field is the minimum value that is added to the section property values and the *MAX* field contains the maximum value that is added to the property values. Hence, the values specified for *MIN* and *MAX* have the physical unit of a length.

VAL

The variability of the section property is described by the modification of the section property value itself. This is only applicable if the section property value is identical and constant for all selected elements. This means that the *MIN* field is the minimum value that the section property value itself can take and the *MAX* field contains the maximum value, which the section property value can take. This is the default if the section property is identical and constant from element to element.

Order

Derivation order. By default, driven by the application.

Notes

Defines the real constant property of a single element or a set of elements as an input variable of the DesignXplorer VT. The real constant property must be specified with the **R** command prior to the **SXREAL** command. The DesignXplorer VT will evaluate the derivatives of the result parameter(s) (see **SXRSLT** command) with respect to the section property defined here. The derivatives are used to determine an approximation function for the result parameter(s) as a function of the section property in the range specified by the *MIN* and *MAX* fields.

Using either the *NSET* field or the *ElComp* field a set of elements is selected for which the real constant property is varied. The user must define one of these two fields and leave the respective other one blank.

If the real constant property is different from element to element, then each element has different values for the real constant properties at the individual nodes. To describe a variation of the real constant property with one single parameter is only possible by using a scalar addition to all real constant property values (*VarType* = ADD) or by using a scalar multiplication factor for all real constant property values (*VarType* = FACT). Varying the real constant property by its value itself using *VarType* = VAL is only possible if the element thickness is constant and identical for all elements.

An error message is generated and the **SXREAL** command is ignored when the current value is outside the variation range.

If the Order is not specified, it will be increased until the accuracy requirements of the **SXRSLT** command are satisfied. When you set a value for Order, no further adjustments for accuracy will be performed. Order should only be used when an input parameter has an influence with a limited order.

Menu Paths

Main Menu>DesignXplorer VT>Setup>Real Constant

SXRFIL, *Fname*, *Ext*, *Dir*

Specifies the file to which DesignXplorer VT results are written.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

Fname
File name (32 characters maximum). Defaults to **Jobname.**

Ext
File name extension (8 characters maximum). Defaults to **rsx** if **Fname** is blank; otherwise, no default.

Dir
Directory name (64 characters maximum). Defaults to the current working directory.

Command Default

Fname = **Jobname**, *Ext* = **rsx**, the default directory is the current working directory.

Notes

Specifies which file is used to store the results of DesignXplorer VT. The file contains the description of the higher order derivatives of the result parameters (selected by the **SXRSLT** command) with respect to the input variables (selected by commands **SXFREQ**, **SXMP**, **SXSEC**,…).

Menu Paths

Main Menu>DesignXplorer VT>Solution>Results File

SXRSLT, *Name*, *Entity*, *Type*, *Comp*, *ACC*, *CompName*

Defines a result quantity for the DesignXplorer VT.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

Name
Name of the result parameter. This is a string of up to 250 characters used for postprocessing purposes. Blanks can be included.

Entity
Entity keyword. Valid keywords are listed in the tables below.

Type
Type of result parameter. Valid labels are listed in the table below depending on the *Entity* field.

Comp
Component of the result parameter type. Valid labels are listed in the table below depending on the *Type* field.

ACC
Required accuracy for the requested result parameter. Default values depend on the individual result parameter. The default value is 0.02 (2%).

CompName
Name of an element or node component, depending on the *Entity* field. If *CompName* = ALL then all entities as specified in the *Entity* field are considered for the **SXRSLT** command. *CompName* = ALL is the default.

Command Default

ACC = 2%, *CompName* = ALL.

Notes

Defines the result quantity, for which an approximation function with respect to the input variables will be calculated.

The accuracy provided with the *ACC* field should not be confused with the error due to meshing of the finite-element model. The accuracy is the maximum difference you can expect between an evaluation of the results of DesignXplorer VT at any point of the input space of the defined input variables and a rerun of a full ANSYS model at the same location of the input space.

The accuracy for all result parameters will be used to derive a global accuracy measure using the minimum value. This means that if there are several result parameters defined with the **SXRSLT** command and some have a requested accuracy of *ACC* = 0.05 and one has a requested accuracy of *ACC* = 0.01, then an accuracy of 0.01 (1%) will be adopted for all result parameters.

If the derivation order has been specified for one parameter, then the accuracy is not guaranteed for any variation involving this parameter.

For a normal modes analysis the result type MASS cannot be selected.

Table of Valid Labels for the *Type* and *Comp* Fields for *Entity*=ELEM

Type	Comp	Description
MASS		Structural mass based on the elements.
S	ALL	Component nodal stress of the element(s) (not averaged at nodes).

Note — For *Entity* = ELEM an existing component name including a set of elements must be specified in the *CompName* field.

Table of valid labels for the *Type* and *Comp* fields for *Entity* = Node

Type	Comp	Description
RF	FX, FY, FZ, MX, MY, MZ, ALL	Nodal reaction forces in the nodal coordinate system

Note — For *Entity* = NODE an existing component name including a set of nodes must be specified in the *CompName* field.

Table of valid labels for the *Type* and *Comp* fields for *Entity* = Mode

Type	Comp	Description
MODE		All mode shapes and the corresponding eigenfrequencies that are evaluated as part of the solution.
FREQ		All eigenfrequencies that are evaluated as part of the solution. Compared to the *Type* = MODE option, the *Type* = FREQ option does not provide the solutions for the mode shapes. The option will save memory compared to the *Type* = MODE option.

Note — For *Entity* = MODE, all modes are always taken into consideration, that is, the only valid user input is *CompName* = ALL.

Menu Paths

Main Menu>DesignXplorer VT>Setup>Result Quantity

SXSEC, *Name*, *MIN*, *MAX*, *RedOpt*, *Lab*, *SECID*, *LAYERID*, *ElComp*, *VarType*, *Order*

Defines a section property as an input variable for the FS Module or the DesignXplorer VT.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

Name
Name for the input variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks can be included.

MIN
Minimum value of the input variable.

MAX
Maximum value of the input variable.

RedOpt
Reduction option that specifies what to do if DesignXplorer VT must reduce the variation range from *MIN* to *MAX*. Valid labels are:

CONT
Continue with the calculation of the derivatives. This is the default.

STOP
Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

Lab
Valid section property label. This label must refer to a valid property of the section as defined by the **SECTYPE** and **SECDATA** commands. The applicable label is:

TK
Shell thickness. This applies only if the section property has been defined with *Type* = SHELL of the **SECTYPE** command.

A
Area of section.

Iyy
Moment of inertia about the y axis.

Iyz
Product of inertia.

Izz
Moment of interia about the z axis.

Iw
Wrapping constant.

J
Torsional constant.

SECID
Section identification number as defined by the **SECTYPE** command and further specified by the **SECDATA** command.

LAYERID
Layer identification number as defined by the **SECDATA** command.

ElComp
Name of an element component or number of a single individual element. If *ElComp* = ALL, then all elements are considered for the **SXSEC** command.

VarType
Label indicating how the variability of a spatially dependent element thickness should be handled. This label is ignored if the shell element thickness is constant for all elements selected by either the *SECID* or the *ElComp* field. Valid labels are:

FACT
The variability of the section property is described by a scale factor that is applied to the section property values. In case the section properties are different from element to element, a scale factor will be applied to all section property values. This means that the *MIN* field is the value of the minimum factor to be applied and the *MAX* field is the maximum factor to be applied. As such the values specified for *MIN* and *MAX* are unitless quantities. *VarType* = FACT is the default if the section property is different from element to element.

ADD
The variability of the section property is described by an add-on that is applied to the section property values. In case the section properties are different from element to element, then the section property values are shifted at the same time by this add-on. This means that the *MIN* field is the minimum value that is added to the section property values and the *MAX* field contains the maximum value that is added to the section property values. Hence, the values specified for *MIN* and *MAX* have the physical unit of a length.

VAL
The variability of the section property is described by the modification of the section property value itself. This is only applicable if the section property value is identical and constant for all selected elements. This means that the *MIN* field is the minimum value that the section property value itself can take and the *MAX* field contains the maximum value, which the section property value can take. This is the default if the section property is identical and constant from element to element.

Order
Derivation order. By default, driven by the application.

Notes

Defines the section property of a single element or a set of elements as an input variable of DesignXplorer VT. The section property must be specified with the **SECTYPE** and **SECDATA** commands prior to the **SXSEC** command. The DesignXplorer VT will evaluate the derivatives of the result parameter(s) (see **SXRSLT** command) with respect to the section property defined here. The derivatives are used to determine an approximation function for the result parameter(s) as a function of the section property in the range specified by the *MIN* and *MAX* fields.

Using either the *SECID* field or the *ElComp* field a set of elements is selected for which the section property is varied. The user must define one of these two fields and leave the respective other one blank.

If the section property is different from element to element, then each element has different values for the section property at the individual nodes. To describe a variation of the section property with one single parameter is only possible by using a scalar addition to all section property values (*VarType* = ADD) or by using a scalar multiplication factor for all section property values (*VarType* = FACT). Varying the section property by its value itself using *VarType* = VAL is only possible, if the section property is constant and identical for all elements.

An error message is generated and the **SXSEC** command is ignored when the current value is outside the variation range.

If the Order is not specified, it will be increased until the accuracy requirements of the **SXRSLT** command are satisfied. When you set a value for Order, no further adjustments for accuracy will be performed.

Menu Paths

Main Menu>DesignXplorer VT>Setup>Section Property

SXSL, *Type*, *Varname*

Selects a subset of elements associated with an SX input variable.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

Type
Label identifying the type of select:

S
Select a new set. This is the default.

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

Varname
Name of an SX input variable, defined with the **SXMP**, **SXREAL**, **SXSEC** or **SXDISC** commands.

Command Default

There are no default values for the command fields.

Notes

This command does not apply to output variables or **SXGEOM** input variables.

Menu Paths

Main Menu> DesignXplorer VT> Setup> Frequency

SXSTAT, *EntyLis*

Print the status of the DesignXplorer VT definitions and settings into a separate window.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

EntyLis
Option that specifies whether the nodes or element lists should be printed for those Variational Technology commands where nodes or element lists have been specified. Valid labels are:

NO
Do not print out the list of nodes or elements (default).

YES
Print out the list of nodes or elements.

Notes

Prints the status the user definitions and settings of the DesignXplorer VT to a separate window.

Menu Paths

Main Menu>DesignXplorer VT>Other>Status

SXTEMP, *Name*, *MIN* , *MAX*, *RedOpt*, *---*, *VarType*, *Order*

Defines the temperature as input variable for the DesignXplorer VT.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

Name
Name for the input variable. This is a string of up to 250 characters used for postprocessing purposes. Blanks are allowed in the string.

MIN
Minimum value of the input variable.

MAX
Maximum value of the input variable.

RedOpt
Reduction option that specifies what to do if the DesignXplorer VT needs to reduce the variation range from *MIN* to *MAX*. Valid labels are:

CONT
Continue with the calculation of the derivatives. This is the default.

STOP
Stop the calculation of the derivatives. This option is necessary if a reduction of the variation range is not acceptable.

Unused field.

VarType
Label indicating how the variability of temperature should be handled. Valid labels are:

FACT
The variability of the material property is described by an scaling factor that is applied to the temperature. This means that the *MIN* field is the value of the minimum factor to be applied and the *MAX* field is the maximum factor to be applied. Hence, the values specified for *MIN* and *MAX* are unitless quantities. *Var-Type* = FACT is the default for temperature parameters.

ADD
The variability of the temperature is described by an add-on that is applied to the temperature values. This means that the *MIN* field is the minimum value that is added to the temperature values and the *MAX* field contains the maximum value that is added to the material property values. Hence, the values specified for *MIN* and *MAX* have the same unit as the material property itself.

Order
Derivation Order. By default, driven by the application.

Command Default

Defaults as outlined above.

Notes

Defines the temperature as an input variable.

If the Order is not specified, it will be increased until the accuracy requirements of the **SXRSLT** command are satisfied. When you set a value for Order, no further adjustments for accuracy will be performed.

Menu Paths

Main Menu>DesignXplorer VT>Setup>Temperature Load

SXVMOD, *Name*, *Oper*

Modifies the status or current value of an input variable for the DesignXplorer VT.

VARIATIONAL TECHNOLOGY: DesignXplorer VT

<> <> <> <> VT <> <> <> <> <> <>

Name
Name for an input or results variable of the DesignXplorer VT. This is a string of up to 250 characters used for postprocessing purposes. Blanks can be used. An input variable with this name must be previously defined using one of the following commands: **SXFREQ**, **SXMP**, **SXRSLT**, …

Oper
Label for the type of operation that is performed on the input variable Name. Valid labels are:

ACT
Activates an input variable in the DesignXplorer VT. This operation is ignored, if the variable specified is already activated.

DEACT
Deactivates an input variable of the DesignXplorer VT. This operation is ignored, if the variable specified is not activated.

DEL
Deletes an input or results variable of the DesignXplorer VT.

Notes

Modifies the status of an input variable for the DesignXplorer VT that have been previously defined, by activating, deactivating or deleting it.

Menu Paths

Main Menu>DesignXplorer VT>Setup>Modify

/SYP, *String*, *ARG1*, *ARG2*, *ARG3*, *ARG4*, *ARG5*, *ARG6*, *ARG7*, *ARG8*

Passes a command string and arguments to the operating system.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP <>

String
Command string (cannot include commas). See also the **/SYS** command.

ARG1, *ARG2*, *ARG3*, *ARG4*, *ARG5*, *ARG6*, *ARG7*, *ARG8*
Arguments to be appended to the command string, separated by blanks, commas, or other delimiter characters (see the *ANSYS Operations Guide*). The arguments may be numbers, parameters, or parametric expressions.

Notes

Passes a command string to the operating system for execution, along with arguments to be appended to the command string. See the *ANSYS Operations Guide* for details. ANSYS may not be aware of your specific user environment. In particular, this command may not recognize UNIX aliases, depending on the hardware platform and user environment.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

/SYS, *String*

Passes a command string to the operating system.

SESSION: Run Controls

MP ME ST DY <> PR EM EH FL PP <>

String
Command string, up to 75 characters (including blanks, commas, etc.). The specified string is passed verbatim to the operating system, i.e., no parameter substitution is performed.

Notes

Passes a command string to the operating system for execution (see the *ANSYS Operations Guide*). Typical strings are system commands such as list, copy, rename, etc. Control returns to the ANSYS program after the system procedure is completed. ANSYS may not be aware of your specific user environment. In particular, this command may not recognize UNIX aliases, depending on the hardware platform and user environment.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

T Commands

TALLOW, *TEMP1*, *TEMP2*, *TEMP3*, *TEMP4*, *TEMP5*, *TEMP6*
Defines the temperature table for safety factor calculations.

POST1: Element Table

MP ME ST DY <> PR <> <> <> PP ED

TEMP1, *TEMP2*, *TEMP3*, *TEMP4*, *TEMP5*, *TEMP6*
Input up to six temperatures covering the range of nodal temperatures. Temperatures must be input in ascending order.

Notes

Defines the temperature table for safety factor calculations [**SFACT**, **SALLOW**]. Use **STAT** command to list current temperature table. Repeat **TALLOW** command to zero table and redefine points (6 maximum).

Safety factor calculations are not supported by PowerGraphics. Both the **SALLOW** and **TALLOW** commands must be used with the Full Model Graphics display method active.

Menu Paths

Main Menu>General Postproc>Safety Factor>Allowable Strs>Reset Temps
Main Menu>General Postproc>Safety Factor>Allowable Strs>Temp-depend

---

TB, *Lab*, *MAT*, *NTEMP*, *NPTS*, *TBOPT*, *EOSOPT*
Activates a data table for nonlinear material properties or special element input.

PREP7: Data Tables

MP ME ST DY <> PR EM EH <> PP ED

Lab
Type of data table:

ANAND
Anand plasticity (VISCO106, VISCO107, VISCO108). See ANAND Specifications for more information.

ANEL
Anisotropic elastic matrix (SOLID5, PLANE13, SOLID64, SOLID98, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SHELL208, SHELL209, and explicit dynamic elements SOLID164 and SOLID168). Also PLANE223, SOLID226, SOLID227. See ANEL Specifications for more information.

ANISO
Anisotropic plasticity (LINK1, PLANE2, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, SOLID65, PLANE82, SHELL91, SOLID92, SHELL93, SOLID95, and SHELL143). See ANISO Specifications for more information.

BH
Magnetic field data (SOLID5, PLANE13, PLANE53, SOLID62, SOLID96, SOLID97, SOLID98, SOLID117). See BH Specifications for more information.

BISO

Bilinear isotropic hardening using von Mises or Hill plasticity (for von Mises plasticity: LINK1, PLANE2, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, SOLID65, PLANE82, SHELL91, SOLID92, SHELL93, SOLID95, SHELL143, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, SHELL209, and explicit dynamic elements PLANE162, SHELL163, SOLID164, and SOLID168. For Hill plasticity: PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). See BISO Specifications for more information.

BKIN

Bilinear kinematic hardening using von Mises or Hill plasticity (For von Mises plasticity: LINK1, PLANE2, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, SOLID65, PLANE82, SHELL91, SOLID92, SHELL93, SOLID95, SHELL143, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, SHELL209, and explicit dynamic elements LINK160, BEAM161, PLANE162, SHELL163, SOLID164, and SOLID168. For Hill plasticity: PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). See BKIN Specifications for more information.

CAST

Cast iron material model (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). See CAST Specifications for more information.

CHABOCHE

Chaboche nonlinear kinematic hardening using von Mises or Hill plasticity (PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). See CHABOCHE Specifications for more information.

COMP

Composite material models (explicit dynamic elements PLANE162, SHELL163, SOLID164, and SOLID168). See COMP Specifications for more information.

CONCR

Concrete element data (SOLID65) or concrete damage model (explicit dynamic elements SOLID164 and SOLID168). See CONCR Specifications for more information.

CREEP

Viscoplasticity/creep. ANSYS can model pure creep, creep with isotropic hardening plasticity, or creep with kinematic hardening plasticity using both von Mises or Hill potentials. See Material Model Combinations in the *ANSYS Elements Reference* for further information on combining models (For explicit creep with von Mises potential: LINK1, PLANE2, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, SOLID65, PLANE82, SOLID92, SOLID95, and SHELL143. For implicit creep with von Mises or Hill potential: PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). See CREEP Specifications for more information.

DISCRETE

Explicit spring-damper (discrete) material models (COMBI165). See DISCRETE Specifications for more information.

DP

Drucker-Prager plasticity (LINK1, PLANE2, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, SOLID65, PLANE82, SHELL91, SOLID92, SHELL93, SOLID95, and SHELL143). See DP Specifications for more information.

DPER

Anisotropic electric permittivity (PLANE223, SOLID226, SOLID227). See DPER Specifications for more information.

EOS
Equation of state (explicit dynamic elements only). See EOS Specifications for more information.

EVISC
Viscoelastic element data (VISCO88, VISCO89, and explicit dynamic elements BEAM161, PLANE162, SOLID164, and SOLID168). See EVISC Specifications for more information.

FAIL
Composite material failure data (SOLID46, SHELL91, SOLID95, SHELL99, SOLID191). See FAIL Specifications for more information. See also the **FC** command (Section 13.2.3: Specifying Failure Criteria of the *ANSYS Structural Analysis Guide*) which applies to all structural shell and solid elements.

FCON
Fluid conductance data (FLUID116). See FCON Specifications for more information.

FOAM
Foam material models (explicit dynamic elements PLANE162, SOLID164, and SOLID168). See FOAM Specifications for more information.

GASKET
Gasket material model (INTER192, INTER193, INTER194, and INTER195). See GASKET Specifications for more information.

GCAP
Geological Cap material model (explicit dynamic elements SOLID164 and SOLID168). See GCAP Specifications for more information.

HFLM
Film coefficient data (FLUID116). See HFLM Specifications for more information.

HILL
Hill anisotropy when combined with other material options simulates plasticity, viscoplasticity, and creep -- all with the Hill potential. See Material Model Combinations in the *ANSYS Elements Reference* for further information on combining models (PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM 188, BEAM189, SHELL208, and SHELL209). See HILL Specifications for more information.

HONEY
Honeycomb material models (explicit dynamic elements PLANE162, SOLID164, and SOLID168). See HONEY Specifications for more information.

HYPER
Hyperelasticity models [Mooney-Rivlin, Ogden, Neo-Hookean, Polynomial form, Arruda-Boyce, Gent, Yeoh, Blatz-Ko, Ogden foam, and user-defined] (SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SHELL208, and SHELL209). See HYPER Specifications for more information.

JOIN
Linear and nonlinear elastic stiffness, linear and nonlinear damping, and hysteretic frictional behavior (MPC184). See JOIN Specifications for more information.

KINH
Multilinear kinematic hardening using von Mises or Hill plasticity (for von Mises plasticity: LINK1, PLANE2, LINK8, PIPE20, BEAM23, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, PLANE82, SOLID92, SHELL93, SOLID95, SHELL143, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209. For Hill plasticity: PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). KINH is the same as MKIN with *TB0PT* = 2, but with less restrictions on the number of points per curve and the number of temperatures. See KINH Specifications for more information.

MELAS
Multilinear elasticity (LINK1, PLANE2, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, SOLID65, PLANE82, SHELL91, SOLID92, SHELL93, SOLID95, and SHELL143). See MELAS Specifications for more information.

MISO
Multilinear isotropic hardening using von Mises or Hill plasticity (for von Mises plasticity: LINK1, PLANE2, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, SOLID65, PLANE82, SHELL91, SOLID92, SHELL93, SOLID95, SHELL143, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209. For Hill plasticity: PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). See MISO Specifications for more information.

MKIN
Multilinear kinematic hardening using von Mises or Hill plasticity (for von Mises plasticity: LINK1, PLANE2, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, SOLID65, PLANE82, SHELL91, SOLID92, SHELL93, SOLID95, SHELL143, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209. For Hill plasticity: PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). See MKIN Specifications for more information.

MOONEY
Mooney-Rivlin hyperelastic element data (HYPER56, HYPER58, HYPER74, HYPER84, HYPER86, HYPER158, and explicit dynamic elements PLANE162, SHELL163, SOLID164, and SOLID168). See MOONEY Specifications for more information.

NLISO
Voce isotropic hardening law for modeling nonlinear isotropic hardening using von Mises or Hill plasticity. (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM 188, BEAM189, SHELL208, and SHELL209). See NLISO Specifications for more information.

PIEZ
Piezoelectric matrix (SOLID5, PLANE13, SOLID98). Also PLANE223, SOLID226, SOLID227. See PIEZ Specifications for more information.

PLAW
Plasticity laws (explicit dynamic elements LINK160, BEAM161, PLANE162, SHELL163, SOLID164, and SOLID168). See PLAW Specifications for more information.

PRONY
Prony series constants for viscoelastic materials (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). See PRONY Specifications for more information.

PZRS
Piezoresistive materials (PLANE223, SOLID226, SOLID227). See PZRS Specifications for more information.

RATE
Rate-dependent plasticity (viscoplasticity) when combined with the BISO, MISO, or NLISO material options, or rate-dependent anisotropic plasticity (anisotropic viscoplasticity) when combined with the HILL and BISO, MISO, or NLISO material options. See Material Model Combinations in the *ANSYS Elements Reference* for further information on combining models (PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). See RATE Specifications for more information.

SHIFT
Shift function for viscoelastic materials (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). See SHIFT Specifications for more information.

SMA
Shape memory alloy for simulation of hysteresis superelastic behavior with no performance deformation. Plane stress is not supported. See Shape Memory Alloys in the *ANSYS Elements Reference*, and Section 4.5: Shape Memory Alloy Material Model in the *ANSYS, Inc. Theory Reference* for further information. The **TB**,SMA option is valid for PLANE182 and PLANE183 (with plane strain or axisymmetric stress states), and for SOLID185, SOLID186, and SOLID187. See SMA Specifications for more information.

STATE
User-defined state variables, used with **TB**,USER or **TB**,CREEP [when the USER CREEP option is chosen (*TBOPT* = 100)]. When used with **TB**,USER, **TB**,STATE is only used with the USERMAT subroutine (*not* USERPL). When used with **TB**,CREEP, **TB**,STATE is only used with the USERCREEP subroutine. See STATE Specifications for more information.

SWELL
Swelling constants (LINK1, PLANE2, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, SOLID65, PLANE82, SOLID92, SOLID95, and SHELL143). See SWELL Specifications for more information.

UNIAXIAL
Uniaxial stress-strain relation associated with the Cast Iron material model (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). See UNIAXIAL Specifications for more information.

USER
User-defined material model; general purpose except for incompressible material models (LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209). Also, user-defined plasticity or viscoplasticity (LINK1, PLANE2, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, SOLID65, PLANE82, SHELL91, SOLID92, SHELL93, SOLID95, and SHELL143). See USER Specifications for more information.

WATER
Water motion table data (PIPE59). See WATER Specifications for more information.

MAT
Material reference number (defaults to 1; maximum equals 100,000).

NTEMP
The number of temperatures for which data will be provided (if applicable). Temperatures are specified on the **TBTEMP** command.

NPTS
For most labels where *NPTS* is defined, it is the number of data points to be specified for a given temperature. Data points are defined with the **TBDATA** or **TBPT** commands.

EOSOPT
Indicates which equation of state model will be used. Used only for explicit dynamics, and only when *Lab* = EOS.

1
Linear polynomial equation of state

2
Gruneisen equation of state

3
Tabulated equation of state

Data Table Specifications

Presented below is a listing of all the labels for the **TB** command that include specific requirements for the *NTEMP*, *NPTS*, and *TBOPT* options, along with links to references where more information can be found, such as required constants, for the particular label.

BKIN Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 6, Maximum = 6

NPTS:
Not used.

TBOPT:
Stress-strain options (not used in an explicit dynamics analysis).

0 --
No stress relaxation with temperature increase (this is not recommended for nonisothermal problems).

1 --
Rice's hardening rule, which takes into account stress relaxation with increasing temperature (default).

References:
Section 2.5.1.1: Bilinear Kinematic Hardening in the *ANSYS Elements Reference*.

Plastic Material Options in the *ANSYS Structural Analysis Guide*.

Nonlinear Inelastic Models in the *ANSYS LS-DYNA User's Guide*.

MKIN Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 5, Maximum = 5

NPTS:
Not used.

TBOPT:
Stress-strain options.

0 --
No stress relaxation with temperature increase (this is not recommended for nonisothermal problems); also produces thermal ratcheting (default).

1 --
Recalculate total plastic strain using new weight factors of the subvolume.

2 --
Scale layer plastic strains to keep total plastic strain constant; agrees with Rice's model (**TB**, BKIN with *TBOPT* = 1). Produces stable stress-strain cycles.

References:
Section 2.5.1.2: Multilinear Kinematic Hardening in the *ANSYS Elements Reference*.

Plastic Material Options in the *ANSYS Structural Analysis Guide*.

KINH Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum = 40

NPTS:
Number of data points to be specified for a given temperature. Default = 20, Maximum = 20

TBOPT:
Use 0 or leave blank to define stress -vs- total strain curve.

Use 4 or enter "PLASTIC" to define stress -vs- plastic strain curve. This option supports only elements LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, and SHELL209.

References:
Section 2.5.1.2: Multilinear Kinematic Hardening in the *ANSYS Elements Reference*.

Plastic Material Options in the *ANSYS Structural Analysis Guide*.

CHABOCHE Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum value of *NTEMP* is such that *NTEMP* x (1 + 2*NPTS*) = 1000

NPTS:
Number of kinematic models to be superposed. Default = 1, Maximum value of *NPTS* is such that *NTEMP* x (1 + 2*NPTS*) = 1000

TBOPT:
Not used.

References:
Section 2.5.1.3: Nonlinear Kinematic Hardening in the *ANSYS Elements Reference*.

Plastic Material Options in the *ANSYS Structural Analysis Guide*.

MISO Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum = 20

NPTS:
Number of data points to be specified for a given temperature. Default = 20, Maximum = 100

TBOPT:
Not used.

References:
Section 2.5.1.5: Multilinear Isotropic Hardening in the *ANSYS Elements Reference*.

Plastic Material Options in the *ANSYS Structural Analysis Guide*.

BISO Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 6, Maximum = 6

NPTS:
Not used.

TBOPT:
Not used.

References:
Section 2.5.1.4: Bilinear Isotropic Hardening in the *ANSYS Elements Reference*.

Plastic Material Options in the *ANSYS Structural Analysis Guide*.

Nonlinear Inelastic Models in the *ANSYS LS-DYNA User's Guide*.

NLISO Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum = 20

NPTS:
Number of data points to be specified for a given temperature. Default = 4, Maximum = 4

TBOPT:
Not used.

References:
Section 2.5.1.6: Nonlinear Isotropic Hardening in the *ANSYS Elements Reference*.

Plastic Material Options in the *ANSYS Structural Analysis Guide*.

HILL Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum = 40

NPTS:
Not used.

TBOPT:
Not used.

References:
Section 2.5.1.8: Hill's Anisotropy in the *ANSYS Elements Reference*.

Plastic Material Options in the *ANSYS Structural Analysis Guide*.

ANISO Specifications

NTEMP:
Not used.

NPTS:
Not used.

TBOPT:
Not used.

References:
Section 2.5.1.7: Anisotropic in the *ANSYS Elements Reference*.

Plastic Material Options in the *ANSYS Structural Analysis Guide*.

DP Specifications

NTEMP:
Not used.

NPTS:
Not used.

TBOPT:
Not used.

References:
Section 2.5.1.9: Drucker-Prager in the *ANSYS Elements Reference*.

Plastic Material Options in the *ANSYS Structural Analysis Guide*.

MELAS Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum = 20

NPTS:
Number of data points to be specified for a given temperature. Default = 20, Maximum = 100

TBOPT:
Not used.

References:
Section 2.5.1.11: Multilinear Elastic in the *ANSYS Elements Reference*.

Multilinear Elasticity in the *ANSYS Structural Analysis Guide*.

USER Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum value of *NTEMP* is such that *NTEMP* x *NPTS* = 1000

NPTS:
Number of data points to be specified for a given temperature. Default = 48, Maximum value of *NPTS* is such that *NTEMP* x *NPTS* = 1000

TBOPT:
Not used.

References:
Section 2.5.1.13: User in the *ANSYS Elements Reference*.

User Defined Material in the *ANSYS Structural Analysis Guide*.

ANAND Specifications

NTEMP:
Not used.

NPTS:
Not used.

TBOPT:
Not used.

References:
Section 2.5.1.10: Anand's Model in the *ANSYS Elements Reference*.

Viscoplasticity in the *ANSYS Structural Analysis Guide*.

RATE Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum value of *NTEMP* is such that *NTEMP* x *NPTS* = 1000

NPTS:
Number of data points to be specified for a given temperature. Default = 2, Maximum value of *NPTS* is such that *NTEMP* x *NPTS* = 1000

TBOPT:
Rate-dependent viscoplasticity options.

PERZYNA --
Perzyna model (default).

PEIRCE --
Peirce model.

References:
Section 2.5.9: Rate-Dependent Plastic (Viscoplastic) Materials in the *ANSYS Elements Reference*.

Viscoplasticity in the *ANSYS Structural Analysis Guide*.

CREEP Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum value of *NTEMP* is such that *NTEMP* x *NPTS* = 1000 for implicit creep and 250 for explicit creep.

NPTS:
Number of data points to be specified for a given temperature. Default = 12 for implicit creep and 72 for explicit creep, Maximum value of *NPTS* is such that *NTEMP* x *NPTS* = 1000 for implicit creep and 250 for explicit creep.

TBOPT:
Creep model options.

0 --

(or Blank) Explicit creep option. Creep model is defined by constants C_6, C_{12}, and C_{66}, through **TBDATA**. See Section 2.5.11.2.1: Primary Explicit Creep Equation for C6 = 0 through Section 2.5.11.2.14: Irradiation Induced Explicit Creep Equation for C66 = 5 in the *ANSYS Elements Reference* for the associated equations. (Applicable to LINK1, PLANE2, LINK8, PIPE20, BEAM23, BEAM24, PLANE42, SHELL43, SOLID45, SHELL51, PIPE60, SOLID62, SOLID65, PLANE82, SOLID92, SOLID95). C_6 = 100 defines the USER CREEP option for explicit creep. You must define the creep law using the subroutine **USERCR.F**. See the *Guide to ANSYS User Programmable Features* for more information.

1 through 13 --

Implicit creep option. See Table 2.4: "Implicit Creep Equations" in the *ANSYS Elements Reference* for a list of available equations. Use **TBTEMP** and **TBDATA** to define temperature-dependent constants. (Applicable to PLANE42, SOLID45, PLANE82, SOLID92, SOLID95, LINK180 , SHELL181, PLANE182, PLANE183, SOLID185, SOLID186 , SOLID187 , BEAM188, BEAM189, SHELL208, and SHELL209).

100 --

USER CREEP option (Applicable ONLY to, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189). for implicit creep. Use in conjunction with **TB**,STATE for defining the number of state variables. You must define the creep law using the subroutine **USERCREEP.F**. See the *Guide to ANSYS User Programmable Features* for more information. Use **TBTEMP** and **TBDATA** to define temperature-dependent constants.

References:

Section 2.5.11: Creep Equations in the *ANSYS Elements Reference*.

Creep in the *ANSYS Structural Analysis Guide*.

SMA Specifications

NTEMP:

Number of temperatures for which data will be provided. Default = 1, maximum = 40.

NPTS:

Not used.

TBOPT:

Not used

References:

Section 2.5.12: Shape Memory Alloys in the *ANSYS Elements Reference*.

Shape Memory Alloys in the *ANSYS Structural Analysis Guide*, and Section 4.5: Shape Memory Alloy Material Model in the *ANSYS, Inc. Theory Reference*.

STATE Specifications

NTEMP:

Not used.

NPTS:

Number of state variables. Maximum = 1000

TBOPT:

Not used.

References:
Section 2.5.1.13: User and Section 2.5.11.1: Implicit Creep Equations in the *ANSYS Elements Reference*.

User Defined Material and Implicit Creep Procedure in the *ANSYS Structural Analysis Guide*.

SWELL Specifications

NTEMP:
Not used.

NPTS:
Not used.

TB0PT:
Not used.

References:
Section 2.5.13: Swelling Equations in the *ANSYS Elements Reference*.

Swelling in the *ANSYS Structural Analysis Guide*.

BH Specifications

NTEMP:
Not used.

NPTS:
Number of data points to be specified. Default = 20, Maximum = 500

TB0PT:
Not used.

References:
Section 2.5.4: Magnetic Materials in the *ANSYS Elements Reference*.

Additional Guidelines for Defining Regional Material Properties and Real Constants in the *ANSYS Low-Frequency Electromagnetic Analysis Guide*.

PIEZ Specifications

NTEMP:
Not used.

NPTS:
Not used.

TB0PT:
Piezoelectric matrix options.

0 --
Piezoelectric stress matrix [e] (used as supplied)

1 --
Piezoelectric strain matrix [d] (converted to [e] form before use)

References:
Section 2.5.6: Piezoelectric Materials in the *ANSYS Elements Reference*.

Piezoelectric Analysis in the *ANSYS Coupled-Field Analysis Guide*.

FAIL Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 6, Maximum = 6

NPTS:
Not used.

TBOPT:
Not used.

References:
Failure Criteria in the *ANSYS Elements Reference*.

Specifying Failure Criteria in the *ANSYS Structural Analysis Guide*.

HYPER Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum value of *NTEMP* is such that *NTEMP* x *NPTS* = 1000

NPTS:
Number of data points to be specified for a given temperature, except for *TBOPT* = MOONEY, where *NPTS* is the number of parameters in the Mooney-Rivlin model (2 [default], 3, 5, or 9).

TBOPT:
Hyperelastic material options.

MOONEY --
Mooney-Rivlin model (default), applicable to elements SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, and SOLID187, SHELL208, and SHELL209. You can choose a 2 parameter Mooney-Rivlin model with *NPTS* = 2 (default), or a 3, 5, or 9 parameter model by setting *NPTS* equal to one of these values.

References:

Section 2.5.2.2: Mooney-Rivlin Hyperelastic Material Constants (TB,HYPER) in the *ANSYS Elements Reference*.

Mooney-Rivlin Hyperelastic Option in the *ANSYS Structural Analysis Guide*.

OGDEN --
Ogden model. For *NPTS*, the default = 1 and the maximum is such that *NTEMP* x *NPTS* x 3 = 1000.

References:

Section 2.5.2.4: Ogden Hyperelastic Material Constants in the *ANSYS Elements Reference*.

Ogden Hyperelastic Option in the *ANSYS Structural Analysis Guide*.

NEO --
Neo-Hookean model. For *NPTS*, the default = 2 and the maximum = 2.

References:

Section 2.5.2.1: Neo-Hookean Hyperelastic Material Constants in the *ANSYS Elements Reference*.

Neo-Hookean Hyperelastic Option in the *ANSYS Structural Analysis Guide*.

POLY --
Polynomial form model. For *NPTS*, the default = 1 and the maximum is such that *NTEMP* x *NPTS* = 1000.

References:

Section 2.5.2.3: Polynomial Form Hyperelastic Material Constants in the *ANSYS Elements Reference*.

Polynomial Form Hyperelastic Option in the *ANSYS Structural Analysis Guide*.

BOYCE--
Arruda-Boyce model. For *NPTS*, the default = 3 and the maximum = 3.

References:

Section 2.5.2.5: Arruda-Boyce Hyperelastic Material Constants in the *ANSYS Elements Reference*.

Arruda-Boyce Hyperelastic Option in the *ANSYS Structural Analysis Guide*.

GENT --
Gent model. For *NPTS*, the default = 3 and the maximum = 3.

References:

Section 2.5.2.6: Gent Hyperelastic Material Constants in the *ANSYS Elements Reference*.

Gent Hyperelastic Option in the *ANSYS Structural Analysis Guide*.

YEOH --
Yeoh model. For *NPTS*, the default = 1 and the maximum is such that *NTEMP* x *NPTS* x 2 = 1000.

References:

Section 2.5.2.7: Yeoh Hyperelastic Material Constants in the *ANSYS Elements Reference*.

Yeoh Hyperelastic Option in the *ANSYS Structural Analysis Guide*.

BLATZ --
Blatz-Ko model. For *NPTS*, the default = 1 and the maximum = 1.

References:

Section 2.5.2.8: Blatz-Ko Foam Hyperelastic Material Constants in the *ANSYS Elements Reference*.

Blatz-Ko Hyperelastic Option in the *ANSYS Structural Analysis Guide*.

FOAM --
Hyperfoam (Ogden) model. For *NPTS*, the default = 1 and the maximum is such that *NTEMP* x *NPTS* x 3 = 1000.

References:

Section 2.5.2.9: Ogden Compressible Foam Hyperelastic Material Constants in the *ANSYS Elements Reference*.

Ogden Compressible Foam Hyperelastic Option in the *ANSYS Structural Analysis Guide*.

USER --
User-defined hyperelastic model. See the *ANSYS Guide to User Programmable Features* for details.

References:

Section 2.5.2.10: User-Defined Hyperelastic Material in the *ANSYS Elements Reference*.

User-Defined Hyperelastic Option in the *ANSYS Structural Analysis Guide*.

MOONEY Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 6, Maximum = 6 (*NTEMP* is not used for explicit dynamic elements)

NPTS:
Not used.

TBOPT:
Mooney-Rivlin material option, applicable to elements HYPER56, HYPER58, HYPER74, HYPER84, HYPER86, HYPER158, SHELL181, SHELL208, and SHELL209, and explicit dynamic elements PLANE162, SHELL163, SOLID164, and SOLID168.

0 --
Direct input of hyperelastic material constants (default).

1 --
Material constants to be determined from experimental data (for use with the ***MOONEY** command). This option is not valid for explicit dynamic elements.

2 --
Material constants to be calculated by the LS-DYNA program from experimental data. This option is only valid for explicit dynamic elements.

References:
Section 2.5.2.11: Mooney-Rivlin Hyperelastic Material Constants (TB,MOONEY) in the *ANSYS Elements Reference*.

Mooney-Rivlin Hyperelastic Option in the *ANSYS Structural Analysis Guide*.

Nonlinear Elastic Models in the *ANSYS LS-DYNA User's Guide*.

WATER Specifications

NTEMP:
Not used.

NPTS:
Not used.

TBOPT:
Not used.

References:
PIPE59 in the *ANSYS Elements Reference*.

GASKET Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1.

NPTS:
Number of data points to be specified for a given temperature. Default = 5 for *TB0PT* = PARA. Default = 1 for all other values of *TB0PT*.

TB0PT:
Gasket material options.

PARA --
Gasket material general parameters.

COMP --
Gasket material compression data.

LUNL --
Gasket linear unloading data.

NUNL --
Gasket nonlinear unloading data.

TSS --
Transverse shear data.

References:
Gasket Materials in the *ANSYS Elements Reference*.

Gasket Joints Simulation in the *ANSYS Structural Analysis Guide*.

ANEL Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 6, Maximum = 6. *NTEMP* is not used for explicit dynamic elements.

NPTS:
Not used.

TB0PT:
Anisotropic elastic matrix options.

0 --
Elasticity matrix used as supplied (input in stiffness form).

1 --
Elasticity matrix inverted before use (input in flexibility form). This option is not valid for explicit dynamic elements.

References:
Section 2.5.5: Anisotropic Elastic Materials in the *ANSYS Elements Reference*.

Linear Elastic Models in the *ANSYS LS-DYNA User's Guide*.

CONCR Specifications

NTEMP:
Number of temperatures for which data will be provided (used only if TBOPT = 0 or 1). Default = 6, Maximum = 6

NPTS:
Not used.

TBOPT:
Concrete material options.

0 or 1 --
General ANSYS concrete option for element SOLID65.

2 --
Concrete damage model for explicit dynamic elements SOLID164 and SOLID168.

References:
SOLID65 in the *ANSYS Elements Reference*.

Concrete Damage Model in the *ANSYS LS-DYNA User's Guide*.

GCAP Specifications

NTEMP:
Not used.

NPTS:
Not used.

TBOPT:
Not used.

References:
Pressure Dependent Plasticity Models in the *ANSYS LS-DYNA User's Guide*.

HFLM Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum = 20

NPTS:
Number of data points to be specified for a given temperature. Default = 1, Maximum = 100

TBOPT:
Not used.

References:
FLUID116 in the *ANSYS Elements Reference*.

FCON Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1, Maximum = 20

NPTS:
Number of data points to be specified for a given temperature. Default = 1, Maximum = 100

TBOPT:
Not used.

References:
FLUID116 in the *ANSYS Elements Reference*.

EVISC Specifications

NTEMP:
Not used.

NPTS:
Not used.

TBOPT:
Not used.

References:
Section 2.5.3: Viscoelastic Material Constants in the *ANSYS Elements Reference*.

Nonlinear Elastic Models in the *ANSYS LS-DYNA User's Guide*.

PLAW Specifications

NTEMP:
Not used.

NPTS:
Not used.

TBOPT:
Plasticity options for explicit dynamics elements (no default - must specify).

1 --
Isotropic/kinematic hardening model.

2 --
Strain rate dependent plasticity model used for metal and plastic forming analyses.

3 --
Anisotropic plasticity model (Barlat and Lian).

4 --
Strain rate dependent plasticity model used for superplastic forming analyses.

5 --
Strain rate dependent isotropic plasticity model used for metal and plastic forming analyses.

6 --
Anisotropic plasticity model (Barlat, Lege, and Brem) used for forming processes.

7 --
Fully iterative anisotropic plasticity model for explicit shell elements only.

8 --
Piecewise linear plasticity model for explicit elements only.

9 --
Elastic-plastic hydrodynamic model for explicit elements only.

10 --
Transversely anisotropic FLD (flow limit diagram) model for explicit elements only.

11 --
Modified piecewise linear plasticity model for explicit shell elements only.

12 --
Elastic viscoplastic thermal model for explicit solid and shell elements only.

References:
Nonlinear Inelastic Models in the *ANSYS LS-DYNA User's Guide*.

Pressure Dependent Plasticity Models in the *ANSYS LS-DYNA User's Guide*.

FOAM Specifications

NTEMP:
Not used.

NPTS:
Not used.

TB0PT:
Foam material options for explicit dynamics elements (no default - must specify).

1 --
Rigid, closed cell, low density polyurethane foam material model.

2 --
Highly compressible urethane foam material model.

3 --
Energy absorbing foam material model.

4 --
Crushable foam material model.

References:
Foam Models in the *ANSYS LS-DYNA User's Guide*.

HONEY Specifications

NTEMP:
Not used.

NPTS:
Not used.

TB0PT:
Not used.

References:
Foam Models in the *ANSYS LS-DYNA User's Guide*.

COMP Specifications

NTEMP:
Not used.

NPTS:
Not used.

TB0PT:
Not used.

References:
Composite Damage Model in the *ANSYS LS-DYNA User's Guide*.

EOS Specifications

NTEMP:
Not used.

NPTS:
Not used.

TB0PT:
Equation of state (explicit dynamics elements only - no default, must specify).

1 --
Johnson-Cook material model - for strain, strain rate, and temperature dependent impact/forming analyses.

2 --
Null material model - for allowing equation of state to be considered without computing deviatoric stresses.

3 --
Zerilli-Armstrong material model - for metal forming processes in which the stress depends on strain, strain rate, and temperature.

4 --
Bamman material model - for metal forming processes with strain rate and temperature dependent plasticity. Does not require an additional equation of state (*E0S0PT* is not used).

5 --
Steinberg material model - for modeling high strain rate effects in solid elements with failure.

References:
Equation of State Models in the *ANSYS LS-DYNA User's Guide*.

DISCRETE Specifications

NTEMP:
Not used.

NPTS:
Not used.

TB0PT:
Explicit spring-damper (discrete) material options.

0 --
Linear elastic spring (translational or rotational elastic spring) (default)

1 --
Linear viscous damper (linear translational or rotational damper)

2 --
Elastoplastic spring (elastoplastic translational or rotational spring with isotropic hardening)

3 --
Nonlinear elastic spring (nonlinear elastic translational or rotational spring with arbitrary force/displacement response moment/rotation dependency)

4 --
Nonlinear viscous damper (nonlinear damping with arbitrary force/velocity response moment/rotational velocity dependency)

5 --
General nonlinear spring (general nonlinear translational or rotational spring with arbitrary loading and unloading definitions)

6 --
Maxwell viscoelastic spring (Maxwell viscoelastic translational or rotational spring)

7 --
Inelastic tension or compression-only spring (inelastic tension or compression only, translational or rotational spring)

References:
Spring-Damper (Discrete) Models in the *ANSYS LS-DYNA User's Guide*.

CAST Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1; Max = 10.

NPTS:
Not used.

TBOPT:
Defines hardening type.

ISOTROPIC --
Specifies cast iron plasticity with isotropic hardening.

References:
Section 2.5.1.12: Cast Iron Plasticity in the *ANSYS Elements Reference*.

UNIAXIAL Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1; Max = 10.

NPTS:
Number of data points to be specified for a given temperature. Default = 20; Max = 20.

TBOPT:
Defines stress-strain relationship for cast iron plasticity.

TENSION --
Defines stress-strain relation in tension

COMPRESSION --
Defines stress-strain relation in compression.

References:
Section 2.5.1.12: Cast Iron Plasticity in the *ANSYS Elements Reference*.

PRONY Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1; Max = 6.

NPTS:
Number of pairs of Prony series. Default = 1 pair; Max = 6 pairs.

TBOPT:
Defines the relaxation behavior for viscoelasticity.

1 --
(or SHEAR) relaxation behavior of the shear response.

2 --
(or BULK) relaxation behavior of the volumetric response.

References:
Section 2.5.3: Viscoelastic Material Constants in the *ANSYS Elements Reference*.

SHIFT Specifications

NTEMP:
Allows one temperature for which data will be provided.

NPTS:
Number of material constants to be entered as determined by the shift function specified by *TBOPT*.

3 --
for *TBOPT* = WLF

2 --
TBOPT = TN

TBOPT:
Defines the shift function

1 --
(or WLF) William-Landel-Ferry shift function.

2 --
(or TN) Tool-Narayanaswamy shift function.

100 --
(or USER) User-defined shift function.

References:
Section 2.5.3: Viscoelastic Material Constants in the *ANSYS Elements Reference*.

PZRS Specifications

NTEMP:
Not used.

NPTS:
Not used.

TB0PT:
Piezoresistive matrix options

0 --
Piezoresistive stress matrix (used as supplied).

1 --
Piezoresistive strain matrix (used as supplied).

References:
Section 2.5.7: Piezoresistive Materials in the *ANSYS Elements Reference*.

Piezoresistive Analysis in the *ANSYS Coupled-Field Analysis Guide*.

DPER Specifications

NTEMP:
Not used.

NPTS:
Not used.

TB0PT:
Permittivity matrix options

0 --
Permittivity matrix at constant strain [ε^S] (used as supplied).

1 --
Permittivity matrix at constant stress [ε^T] (converted to [ε^S] form before use).

References:
Section 2.5.8: Anisotropic Electric Permittivity Materials in the *ANSYS Elements Reference*.

JOIN Specifications

NTEMP:
Number of temperatures for which data will be provided. Default = 1.

NPTS:
Number of data points to be specified for a given temperature. *NPTS* is ignored if *TB0PT* = STIF, DAMP, or FRIC.

TB0PT:
Joint element material options.

Linear behavior:

STIF --
Linear stiffness.

DAMP --
Linear damping.

Nonlinear behavior:

JNSA --
Nonlinear stiffness behavior in all available components of relative motion for the joint element.

JNS4 --
Nonlinear stiffness behavior in local ROTX direction only.

JNS6 --
Nonlinear stiffness behavior in local ROTZ direction only.

JNDA --
Nonlinear damping behavior in all available components of relative motion for the joint element.

JND4 --
Nonlinear damping behavior in local ROTX direction only.

JND6 --
Nonlinear damping behavior in local ROTZ direction only.

Hysteretic friction:

FRIC --
Stick stiffness for hysteretic frictional behavior.

JNFA --
Hysteretic frictional behavior in all available components of relative motion for the joint element.

JNF4 --
Hysteretic frictional behavior in local ROTX direction only.

JNF6 --
Hysteretic frictional behavior in local ROTZ direction only.

References:
Section 2.5.14: MPC184 Joint Materials in the *ANSYS Elements Reference*.

Notes

TB activates a data table to be used with subsequent **TBDATA** or **TBPT** commands. The table space is initialized to zero values. Data from this table are used for certain nonlinear material descriptions as well as for special input for some elements. See Section 2.5: Data Tables - Implicit Analysis in the *ANSYS Elements Reference* for a description of table types (*Lab*) or the elements that require the table for special data. See Material Models in the *ANSYS LS-DYNA User's Guide* for a description of data table input required for explicit dynamic materials. See the **MP** command for linear material property input.

The type of data table specified remains active until the **TB** command is reissued. More than one type of data table may be defined for each material (e.g., MISO and CREEP), except that only one type of plasticity/elasticity may be used for each material.

This command is also valid in SOLUTION.

Product Restrictions

Lab = BH is allowed in ANSYS only with the Emag. Only *Lab* = FAIL is allowed in ANSYS Professional. Only *Lab* = BH is allowed in ANSYS Emag. *Lab* = COMP is not allowed in ANSYS ED.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

TBCOPY, *Lab*, *MATF*, *MATT*

Copies a data table from one material to another (see Notes).

PREP7: Data Tables

MP ME ST DY <> PR EM <> <> PP ED

Lab
Data table label (see the **TB** command for valid labels, and see Notes for *Lab* = ALL).

MATF
Material reference number where data table is to be copied from.

MATT
Material reference number where data table is to be copied to.

Notes

The **TBCOPY** command, with *Lab* = ALL, copies all of the nonlinear data defined by the **TB** command. If you copy a model that includes both yield behavior constants and linear constants (for example, a BKIN model), **TBCOPY**,ALL and **MPCOPY** are used together to copy the entire model. All input data associated with the model is copied, that is, all data defined through the **TB** and **MP** commands.

Also, if you copy a material model using the Material Model Interface (**Edit> Copy**), both the commands **TBCOPY**,ALL and **MPCOPY** are issued, regardless of whether the model includes linear constants only, or if it includes a combination of linear and yield behavior constants.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

TBDATA, *STLOC*, *C1*, *C2*, *C3*, *C4*, *C5*, *C6*

Defines data for the data table.

PREP7: Data Tables

MP ME ST DY <> PR EM <> <> PP ED

STLOC

Starting location in table for entering data. For example, if *STLOC* = 1, data input in the *C1* field applies to the first table constant, *C2* applies to the second table constant, etc. If *STLOC*=5, data input in the *C1* field applies to the fifth table constant, etc. Defaults to the last location filled + 1. The last location is reset to 1 with each **TB** or **TBTEMP** command.

C1, *C2*, *C3*, *C4*, *C5*, *C6*

Data values assigned to six locations starting with *STLOC*. If a value is already in this location, it is redefined. A blank value leaves the existing value unchanged.

Notes

Defines data for the table specified on the last **TB** command at the temperature specified on the last **TBTEMP** command (if applicable). The type of data table specified in the last **TB** command determines the number of data values needed in **TBDATA**. See Section 2.5: Data Tables - Implicit Analysis of the *ANSYS Elements Reference* for the number of data values required for different material behavior options.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

TBDELE, *Lab*, *MAT1*, *MAT2*, *INC*

Deletes previously defined data tables.

PREP7: Data Tables

MP ME ST DY <> PR EM <> <> PP ED

Lab

Data table label (see the **TB** command for valid labels). If ALL, delete all data tables.

MAT1, *MAT2*, *INC*

Delete tables for materials *MAT1* to (*MAT2* defaults to *MAT1*) in steps of *INC* (defaults to 1). If *MAT1*= ALL, ignore *MAT2* and *INC* and delete data tables for all materials.

Notes

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

TBFT,*Oper*, *ID*, *Option1*, *Option2*, *Option3*, *Option4*, *Option5*, *Option6*, *Option7*
Performs material curve fitting operations.

PREP7: Materials

MP ME ST <> <> PR EM <> FL PP ED

Use material curve fitting to evaluate experimental data for use as coefficients for certain nonlinear material models built into ANSYS. With this feature, you compare experimental stress data versus ANSYS-calculated stress data for nonlinear models associated with the material behaviors listed below:

For hyperelastic material behavior, see TBFT Specifications for Hyperelastic Models, below.

For creep material behavior, see TBFT Specifications for Creep Models, below.

For viscoelastic material behavior, see TBFT Specifications for Viscoelastic Models, below.

TBFT Specifications for Hyperelastic Models

TBFT,*Oper*, *ID*, *Option1*, *Option2*, *Option3*, *Option4*, *Option5*, *Option6*, *Option7*

Oper
The specific curve fitting operation:

FADD
Define a constitutive model.

FDEL
Delete a constitutive model.

FSET
Write data related to a constitutive model to the database (same as **TB** command).

SET
Initialize coefficients of a constitutive model for nonlinear curve-fitting procedure.

SOLVE
Solve for coefficients.

FIX
Fix (hold constant) the coefficient you specify in *Option4*.

EADD
Add experimental data.

EDEL
Delete experimental data.

LIST
List all data associated with the material model represented by the material *ID* number.

PLOT
Plot curves.

ID

The material reference number (same as *MAT* argument used in the **TB** command). Valid entry is any number greater than zero (default = 1) but less than 100,000.

Option1

For coefficient operations (*Oper* = FADD, FDEL, FSET, SET, SOLVE or FIX) this field specifies the category (HYPER).

For adding, deleting or plotting your experiment (*Oper* = EADD, EDEL, or PLOT), this field specifies the experimental data type. Valid entries are: UNIA, BIAX, SHEA, and VOLU (default = UNIA).

Option2

For coefficient operations (*Oper* = FADD, FDEL, FSET, SET, SOLVE, or FIX), this field specifies constitutive model type. The valid entries are listed in Hyperelastic Options below.

When you need to specify a filename from which to get experimental data (*Oper* = EADD), place that string here. Valid entry is any filename string. You can enter the entire **path\filename.extension** string and leave the next two fields (*Option3* and *Option4*) blank, or you can specify the name here, the extension in the next field, and the path following.

For plotting (*Oper* = PLOT), enter HYPER.

Option3

For *Oper* = FADD, FDEL, FSET, SET, SOLVE or FIX, some of the cases specified in*Option2* will require that the polynomial order be specified. The applicable values for the order specification are listed in Hyperelastic Options

If a filename for experimental data is being specified in *Option2* (*Oper* = EADD), this field will contain the file extension.

For plotting operations (*Oper* = PLOT), use this field to specify the constitutive model name (see Hyperelastic Options).

Option4

When you are working on a specific coefficient (*Oper* = SET or FIX), this field , specifies the index of that coefficient. Valid entries vary from 1 to N, where N is the total number of coefficients (default = 1).

If a filename for experimental data is being specified in *Option2* (*Oper* = EADD), this field will contain the directory/path specification.

For *Oper* = PLOT, this field specifies the order of the polynomial if applicable, see Hyperelastic Options .

If *Oper* = SOLVE, this value specifies the curve-fitting procedure. Valid entries are 0 for unnormalized least squares curve-fitting procedure, and 1 for normalized least squares curve-fitting procedure.

Option5

If you are defining your coefficients (*Oper* = SET), this field specifies the initial value of your coefficient.

If *Oper* = SOLVE, use this field to specify the number of iterations to be used in the calculation of the coefficients. Valid entry is any positive integer (default = 1000)

If you specify a coefficient to be held constant (*Oper* = FIX), enter a 1 to FIX the coefficient, and a 0 to allow it to vary (turn fixing OFF).

Option6

If *Oper* = SOLVE, specifies the allowed tolerance in residual change to stop an iteration. Valid entry is 0.0 to 1.0 (default = 0.0).

Option7

If *Oper* = SOLVE, specifies the allowed tolerance in coefficient change to stop an iteration. Valid entry is 0 to 1 (default = 0).

Hyperelastic Options

Option1	Option2	Option3
HYPER	MOON	2, 3, 5, 9
HYPER	POLY	1 to N
HYPER	OGDEN	1 to N
HYPER	YEOH	1 to N
HYPER	BOYC	NA
HYPER	GENT	NA
HYPER	NEO	NA
HYPER	BLAT	NA
HYPER	FOAM	1 to N

Following table summarizes the format for hyperelastic operations via the **TBFT** command:

Hyperelastic Model Command Summary

Oper	ID	Option1	Option2	Option3	Option4	Option5	Option6	Option7
FADD	ID	HYPER	Option	Order				
FDEL	ID	HYPER	Option	Order				
FSET	ID	HYPER	Option	Order				
SET	ID	HYPER	Option	Order	Index	Value		
SOLVE	ID	HYPER	Option	Order	Norm Flag	Num Iter	RTOL	CTOL
FIX	ID	HYPER	Option	Order	Index	Fix / Unfix Flag		
EADD	ID	Exp Type	File	Extension	Directory			
EDEL	ID	Exp Type / Index						
PLOT	ID	Exp Type / Index	HYPER	Option	Order			

TBFT Specifications for Creep Models

TBFT,*Oper*, *ID*, *Option1*, *Option2*, *Option3*, *Option4*, *Option5*, *Option6*, *Option7*

Oper

The specific curve fitting operation:

FADD

Define a constitutive model.

FDEL
Delete a constitutive model.

FSET
Write data related to a constitutive model to the database (same as **TB** command).

SET
Initialize coefficients of a constitutive model for nonlinear curve-fitting procedure.

SOLVE
Solve for coefficients.

FIX
Fix (hold constant) the coefficient you specify in *Option4*.

EADD
Add experimental data.

EDEL
Delete experimental data.

LIST
List all data associated with the material model represented by the material *ID* number.

PLOT
Plot curves.

ID
The material reference number (same as *MAT* argument used in the **TB** command). Valid entry is any number greater than zero (default = 1) but less than 100,000.

Option1
For coefficient operations (*Oper* = FADD, FDEL, FSET, SET, SOLVE or FIX) this field specifies the category (CREEP).

For adding, deleting or plotting your experiment (*Oper* = EADD, EDEL, or PLOT), this field specifies the experimental data type (CREEP).

Option2
For coefficient operations (*Oper* = FADD, FDEL, FSET, SET, SOLVE, or FIX), this field specifies constitutive model type. The valid entries are listed in Creep Options below.

When you need to specify a filename from which to get experimental data (*Oper* = EADD), place that string here. Valid entry is any filename string. You can enter the entire **path\filename.extension** string and leave the next two fields (*Option3* and *Option4*) blank, or you can specify the name here, the extension in the next field, and the path following.

For plotting (*Oper* = PLOT), enter HYPER.

Option3
If a filename for experimental data is being specified in *Option2* (*Oper* = EADD), this field will contain the file extension.

For plotting operations (*Oper* = PLOT), use this field to specify the constitutive model name (see Creep Options).

Option4
When you are working on a specific coefficient (*Oper* = SET or FIX), this field , specifies the index of that coefficient. Valid entries vary from 1 to N, where N is the total number of coefficients (default = 1).

If a filename for experimental data is being specified in *Option2* (*Oper* = EADD), this field will contain the directory/path specification.

If *Oper* = SOLVE, this value specifies the curve-fitting procedure. Valid entries are 0 for unnormalized least squares curve-fitting procedure, and 1 for normalized least squares curve-fitting procedure.

Option5

If you are defining your coefficients (*Oper* = SET), this field specifies the initial value of your coefficient.

If *Oper* = SOLVE, use this field to specify the number of iterations to be used in the calculation of the coefficients. Valid entry is any positive integer (default = 1000)

If you specify a coefficient to be held constant (*Oper* = FIX), enter a 1 to FIX the coefficient, and a 0 to allow it to vary (turn fixing OFF).

Option6

If *Oper* = SOLVE, specifies the allowed tolerance in residual change to stop an iteration. Valid entry is 0.0 to 1.0 (default = 0.0).

Option7

If *Oper* = SOLVE, specifies the allowed tolerance in coefficient change to stop an iteration. Valid entry is 0 to 1 (default = 0).

Creep Options

Category	Name	Option
CREEP	SHAR	NA
CREEP	THAR	NA
CREEP	GEXP	NA
CREEP	GGRA	NA
CREEP	GBLA	NA
CREEP	MTHA	NA
CREEP	MSHA	NA
CREEP	GGAR	NA
CREEP	EXPO	NA
CREEP	NORT	NA
CREEP	PSTH	NA
CREEP	PSRP	NA
CREEP	GTHA	NA

Following table summarizes the format for creep operations via the **TBFT** command:

Creep Model Command Summary

Oper	ID	Option1	Option2	Option3	Option4	Option5	Option6	Option7
FADD	ID	CREEP	Option	NA				
FDEL	ID	CREEP	Option	NA				
FSET	ID	CREEP	Option	NA				
SET	ID	CREEP	Option	NA	Index	Value		

Oper	ID	Option1	Option2	Option3	Option4	Option5	Option6	Option7
SOLVE	ID	CREEP	Option	NA	Norm Flag	Num Iter	RTOL	CTOL
FIX	ID	CREEP	Option	NA	Index	Fix / Unfix Flag		
EADD	ID	Exp Type	File	Extension	Directory			
EDEL	ID	Exp Type / Index						
PLOT	ID	Exp Type / Index	CREEP	Option				

TBFT Specifications for Viscoelastic Models

TBFT,*Oper*, *ID*, *Option1*, *Option2*, *Option3*, *Option4*, *Option5*, *Option6*, *Option7*

Oper
The specific curve fitting operation:

FCASE
Define a case/constitutive model for viscoelasticity (prony).

FADD
Define a constitutive model.

FDEL
Delete a constitutive model.

FSET
Write data related to a constitutive model to the database (same as **TB** command).

SET
Initialize coefficients of a constitutive model for nonlinear curve-fitting procedure.

SOLVE
Solve for coefficients.

FIX
Fix (hold constant) the coefficient you specify in *Option4*.

EADD
Add experimental data.

EDEL
Delete experimental data.

LIST
List all data associated with the material model represented by the material *ID* number.

PLOT
Plot curves.

ID
The material reference number (same as *MAT* argument used in the **TB** command). Valid entry is any number greater than zero (default = 1) but less than 100,000.

Option1

This option is set to VISCO for adding coefficients (*Oper* = FADD). The FADD operation is used to specify the shear order, the bulk order or the shift option.

This option (*Oper* = CASE) is set to NEW or FINI. The FADD commands are always enclosed by **TBFT**,FCASE, ID, NEW and **TBFT**,FCASE, ID, FINI. See Chapter 9, "Material Curve Fitting" in the *ANSYS Structural Analysis Guide* for more information.

For other coefficient operations (*Oper* = FDEL, FSET, SET, SOLVE or FIX) this field is set to CASE.

For adding, deleting or plotting your experiment (*Oper* = EADD, EDEL, or PLOT), this field specifies the experiment type. Valid entries are: SDEC (Shear Modulus vs. Time) or BDEC (Bulk Modulus vs. Time).

Option2

For defining your viscoelastic case (*Oper* = FCASE), you specify PVHE.

For coefficient operations (*Oper* = FDEL, FSET, SET, SOLVE, or FIX), this field specifies the case name.

To create a new case (*Oper* = FADD), the valid options are listed in Viscoelastic Options below.

When you need to specify a filename from which to get experimental data (*Oper* = EADD), place that string here. Valid entry is any filename string. You can enter the entire **path\filename.extension** string and leave the next two fields (*Option3* and *Option4*) blank, or you can specify the name here, the extension in the next field, and the path following.

For plotting (*Oper* = PLOT), enter the CASE name.

Option3

For *Oper* = FCASE, this field specifies the CASE name.

For *Oper* = FADD, this field specifies the order

If a filename for experimental data is being specified in *Option2* (*Oper* = EADD), this field will contain the file extension.

For plotting operations (*Oper* = PLOT), use this field to specify the CASE name.

Option4

When you are working on a specific coefficient (*Oper* = SET or FIX), this field specifies the index of that coefficient. Valid entries vary from 1 to N, where N is the total number of coefficients (default = 1).

You can also specify TREF to indicate the reference temperature, or COMP for a partial/complete solution (only for bulk, only for shear, or all coefficients).

If a filename for experimental data is being specified in *Option2* (*Oper* = EADD), this field will contain the directory/path specification.

If *Oper* = SOLVE, this value specifies the curve-fitting procedure. Valid entries are 0 for unnormalized least squares curve-fitting procedure, and 1 for normalized least squares curve-fitting procedure.

Option5

If *Oper* = SET, this field specifies the initial value of your coefficient according to the coefficient index specified in *Option4*. It will be the reference temperature for *Option4* = TREF. It will be PBULK, PSHEA or PVHE if *Option4* = COMP.

If *Oper* = SOLVE, use this field to specify the number of iterations to be used in the calculation of the coefficients. Valid entry is any positive integer (default = 1000).

If you are specifying a coefficient to be held constant (*Oper* = FIX), a 1 FIXES the specified coefficient, and a 0 allows it to vary (turns fixing OFF).

Option6

If *Oper* = SOLVE, specifies the allowed tolerance in residual change to stop an iteration. Valid entry is 0.0 to 1.0 (default = 0.0).

Option7

If *Oper* = SOLVE, specifies the allowed tolerance in coefficient change to stop an iteration. Valid entry is 0 to 1 (default = 0).

Viscoelastic Options

Category	Name	Option
VISCO	PSHEAR	1 to N
VISCO	PBULK	1 to N
VISCO	SHIFT	TN, WLF, NONE

Following table summarizes the format for viscoelastic operations via the **TBFT** command:

Viscoelastic Models

Oper	ID	Option1	Option2	Option3	Option4	Option5	Option6	Option7
FCASE	ID	NEW	PVHE	Case Name				
FCASE	ID	FINI						
FADD	ID	VISCO	Option	Order				
FDEL	ID	CASE	Case Name	NA				
FSET	ID	CASE	Case Name	NA				
SET	ID	CASE	Case Name	NA	Index	Value		
SOLVE	ID	CASE	Case Name	NA	Norm Flag	Num Iter	RTOL	CTOL
FIX	ID	CASE	Case Name	NA	Index	Fix / Unfix Flag		
EADD	ID	Exp Type	File	Extension	Directory			
EDEL	ID	Exp Type / Index						
PLOT	ID	Exp Type / Index	CASE	Case Name	NA			

Notes

This command provides tools for comparing experimental material data to the calculated data ANSYS supplies for various non linear material options. Based on curve fitting plot comparisons and error norms, you choose the model to use in solution according to the best fit. All of the capabilities of the TBFT-series of commands are accessible interactively via the standard material GUI. See Chapter 9, "Material Curve Fitting" in the *ANSYS Structural Analysis Guide* for more information.

You can display material model data associated with both the **TB** command and the **TBFT**,FSET command by issuing **TBLIST**,ALL,ALL.

Material model data associated with the *latest* **TB** or **TBFT**,FSET command overwrites previous data.

You can display material model data associated with both the **TB** command and the **TBFT**,FSET command by issuing **TBLIST**,ALL,ALL.

The uniaxial, biaxial, and shear experimental data use *engineering* stress. The volumetric data uses *true* stress. See the *ANSYS Elements Reference* for details on experimental data for creep and viscoelasticity.

Menu Paths

Main Menu> Preprocessor> Material Props> Material Models

TBLE
Specifies "Data table properties" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and will appear in the log file (**Jobname**.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Data Tables

TBLIST, *Lab*, *MAT*
Lists the data tables.

PREP7: Data Tables

MP ME ST DY <> PR EM <> <> PP ED

Lab
Data table label (see the **TB** command for labels). Defaults to the active table. If ALL, list data for all labels.

MAT
Material number to be listed (defaults to the active material). If ALL, list data tables for all materials.

Notes

This command is a utility command, valid anywhere.

Menu Paths

Utility Menu>List>Properties>Data Tables

TBMODIF, *ROW*, *COL*, *VALUE*

Modifies data for the data table (GUI).

PREP7: Data Tables

MP ME ST DY <> PR EM <> <> PP ED

ROW, *COL*
The row and column numbers of the table entry to be modified.

VALUE
The new value to be used in the *ROW*, *COL* location.

Notes

Modifies data for the table specified on the last **TB** command. For temperature-dependent data, the temperature specified on the last **TBTEMP** command is used. This is a command generated by the Graphical User Interface (GUI). It will appear in the log file (**Jobname**.LOG) if a **TB** data table is graphically edited in spreadsheet fashion. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

TBPLOT, *Lab*, *MAT*, *TBOPT*, *TEMP*, *SEGN*

Displays the data table.

PREP7: Data Tables

MP ME ST DY <> PR EM <> <> PP ED

Lab
Data table label. Valid labels are: MKIN, KINH, MELAS, MISO, BKIN, BISO, BH, GASKET, and JOIN. Defaults to the active table label. For B-H data, also valid are: NB to display NU-B^2, MH to display MU vs. H, and SBH, SNB, SMH to display the slopes of the corresponding data.

MAT
Material number to be displayed (defaults to the active material).

TBOPT
Gasket material or joint element material option to be plotted.

The following gasket material options are valid only when *Lab* = GASKET:

ALL
Plots all gasket data.

COMP
Plots gasket compression data only.

LUNL
Plots gasket linear unloading data with compression curve.

NUNL
Plots gasket nonlinear unloading data only.

The following joint element material options are valid only when *Lab* = JOIN:

JNSA
Plots nonlinear stiffness data that is applicable to all relevant directions.

JNS*n*
Plots only the specified nonlinear stiffness data. The "*n*" can be 4 or 6. For example, JNS4 plots only the nonlinear stiffness data specified in the local direction 4 (ROTX).

JNDA
Plots nonlinear damping data that is applicable to all relevant directions.

JND*n*
Plots only the specified nonlinear damping data. The "*n*" can be 4 or 6. For example, JND4 plots only the nonlinear damping data specified in the local direction 4 (ROTX).

JNFA
Plots nonlinear hysteretic friction data that is applicable to all relevant directions.

JNF*n*
Plots only the specified nonlinear hysteretic friction data. The "*n*" can be 4 or 6. For example, JNF4 plots only the nonlinear hysteretic friction data specified in local direction 4 (ROTX).

TEMP
Specific temperature at which gasket data or joint element material data will be plotted (used only when *Lab* = GASKET or JOIN). Use *TEMP* = ALL to plot gasket data or joint element material data at all temperatures.

SEGN
Segment number of plotted curve (valid only when *Lab* = GASKET):

NO
Segment number is not added to plotted curve (default).

YES
Segment number is added to plotted curve. This option is ignored if the number of data points in a curve exceeds 20.

Notes

Only data for stress-strain, B-H, gasket curves, or joint element nonlinear material model curves may be displayed. *TBOPT* and *TEMP* are used only when *Lab* = GASKET or JOIN. *SEGN* is used only when *Lab* = GASKET.

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Data Tables

TBPT, *Oper*, *X*, *Y*

Defines a point on a nonlinear data curve.

PREP7: Data Tables

MP ME ST <> <> PR EM <> <> PP ED

Oper

Operation to perform:

DEFI

Defines a new data point (default). The point is inserted into the table in ascending order of *X*. If a point already exists with the same *X* value, it is replaced.

DELE

Deletes an existing point. The *X* value must match the *X* value of the point to be deleted (*Y* is ignored).

X

The *X* value of the point (for example, strain, *H*, etc.).

Y

The corresponding *Y* value of the point (for example, stress, *B*, etc.).

Notes

TBPT defines a point on a nonlinear data curve (such as a stress-strain curve, B-H curve, etc.) at the temperature specified on the last **TBTEMP** command. The meaning of the *X* and *Y* values will depend on the type of data table specified on the last **TB** command (MISO, BH, etc.).

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

TBTEMP, *TEMP*, *KMOD*

Defines a temperature for the data table.

PREP7: Data Tables

MP ME ST DY <> PR <> <> <> PP ED

TEMP

Temperature value (defaults to 0.0 if *KMOD* is blank).

KMOD

If blank, *TEMP* defines a new temperature. If an integer, 1 to NTEMP (from the **TB** command), modify that previously defined temperature to the *TEMP* value, unless *TEMP* is blank, then that previously defined temperature is reactivated. Use **TBLIST** to list temperatures and data. The next **TBDATA** or **TBPT** commands also add or change the data at this temperature. If *KMOD* = CRIT (and *TEMP* is blank), the next **TBDATA** values are failure criteria keys as described for SOLID46, SHELL91, SHELL99, and SOLID191. If *KMOD* = STRAIN (and *TEMP* is blank), the next **TBDATA** values are strains as described for the MKIN property option (see Section 2.5: Data Tables - Implicit Analysis of the *ANSYS Elements Reference*).

Notes

Defines a temperature to be associated with the data on the subsequent **TBPT** or **TBDATA** commands. The temperature remains active until the next **TBTEMP** command is input. Data values must be defined with the temperatures in ascending order. Temperatures previously associated with a data table may also be modified.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

TCHG, *ELEM1*, *ELEM2*, *ETYPE2*

Converts 20-node degenerate tetrahedral elements to their 10-node non-degenerate counterparts.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

ELEM1

Name (or the number) of the 20-node tetrahedron element that you want to convert. This argument is required.

ELEM2

Name (or the number) of the 10-node tetrahedron element to which you want to convert the *ELEM* elements. This argument is required.

ETYPE2

Element TYPE reference number for *ELEM2*. If *ETYPE2* is 0 or is not specified, ANSYS chooses the element TYPE reference number for *ELEM2*. See the Notes section for details. This argument is optional.

Notes

The **TCHG** command allows you to specify conversion of *any* 20-node brick to *any* 10-node tetrahedron. (However, not all combinations make sense.)

The **TCHG** command is useful when used in with the **MOPT**,PYRA command. Twenty-node pyramid shaped elements may be used in the same volume with 10-node tetrahedra.

Performing a conversion is likely to create circumstances in which more than one element type is defined for a single volume.

If specified, *ETYPE2* will usually be the same as the local element TYPE number (**ET**,*ITYPE*) that was assigned to *ELEM2* with the **ET** command. You can specify a unique number for *ETYPE2* if you prefer. Although *ETYPE2* is optional, it may be useful when two or more *ITYPE*s have been assigned to the same element (for example, if two SOLID92 elements have been established in the element attribute tables for the current model, use the *ETYPE2* argument to distinguish between them). If *ETYPE2* is nonzero and it has *not* already been assigned to an element via **ET**, ANSYS assigns the *ETYPE2* value to *ELEM2* as its element TYPE reference number.

If *ETYPE2* is 0 or is not specified, ANSYS determines the element TYPE reference number for *ELEM2* in one of these ways:

- If *ETYPE2* is 0 or is not specified, and *ELEM2 does not appear* in the element attribute tables, ANSYS uses the next available (unused) location in the element attribute tables to determine the element TYPE reference number for *ELEM2*.
- If *ETYPE2* is 0 or is not specified, and *ELEM2 appears* in the element attribute tables, ANSYS uses *ELEM2* 's existing element TYPE reference number for *ETYPE2* . (If there is more than one occurrence of *ELEM2* in the element attribute tables (each with its own TYPE reference number), ANSYS uses the first *ELEM2* reference number for *ETYPE2* .)

See Meshing Your Solid Model of the *ANSYS Modeling and Meshing Guide* for detailed information about converting degenerate tetrahedral elements.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Change Tets

TEE, *NCENT*, *TYPE*, *ELEM*, *EINC*, *L1*, *L2*, *L3*

Defines a tee in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

NCENT
Node where three straight pipes intersect forming a tee (or "Y"). Defaults to last starting branch node [**BRANCH**].

TYPE
Type of tee:

WT
Welding tee (default).

UFT
Unreinforced fabricated tee.

ELEM
Element number to be assigned to first tee leg (defaults to the previous maximum element number (MAXEL) + 1).

EINC
Element number increment (defaults to 1).

L1, *L2*, *L3*
Tee leg lengths (corresponding in order of increasing straight pipe element numbers). Must be less than the straight pipe length. Defaults to 2 x OD of straight pipe (for each leg).

Notes

Defines a tee in place of the tee intersection of three previously defined straight pipe elements. See the PREP7 **RUN** command. The new tee is also composed of three PIPE16 straight pipe elements, but of the leg lengths specified and with the appropriate tee factors calculated. Three new nodes are generated at the ends of the tee. The original three straight pipes are automatically "shortened" to meet the ends of the tee. The tee specifications and loadings are taken from the corresponding three straight pipes.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Define Pipes>Pipe Tee

---

TERM, *Kywrd*, *Opt1*, *Opt2*, *Opt3*

Specifies various terminal driver options.

DISPLAY: Driver Options

MP ME ST DY <> PR EM <> FL PP ED

If *Kywrd* = COPY, command format is TERM,COPY,*NCOPY*

NCOPY
Activate hard copy device for *NCOPY* (0,1,2, etc.) copies.

If *Kywrd* = LOOP, command format is TERM,LOOP,*NLOOP*,*PAUSE*. Used only with PLOT,ALL

NLOOP
Loop *NLOOP* times back to beginning of file when end of file is reached.

PAUSE
Pause *PAUSE* seconds between plots.

If *Kywrd* = NOPROM, command format is TERM,NOPROM,*KEY*

KEY
Prompt key:

0
Display prompt line for prompt.

1
Use terminal bell for prompt.

Notes

Used only with terminal driver names on **/SHOWDISP** command.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

THOPT, *Refopt*, *REFORMTOL*, *NTABPOINTS*, *TEMPMIN*, *TEMPMAX*

Nonlinear transient thermal solution option.

SOLUTION: Analysis Options

MP ME <> <> <> PR <> <> <> PP ED

Refopt

Matrix reform option.

FULL

Use the full Newton-Raphson solution option (default).

QUASI

Use a selective reform solution option based on *REFORMTOL*.

LINEAR

Use a linear solution option with no matrix reform.

REFORMTOL

Property change tolerance for Matrix Reformation (.05 default). The thermal matrices are reformed if the maximum material property change in an element (from the previous reform time) is greater than the reform tolerance.

NTABPOINTS

Number of points in Fast Material Table (64 default).

TEMPMIN

Minimum temperature for Fast Material Table. Defaults to the minimum temperature defined by the **MPTEMP** command for any material property defined.

TEMPMAX

Maximum temperature for Fast Material Table. Defaults to the maximum temperature defined by the **MPTEMP** command for any material property defined.

Notes

This solution logic is only supported by the ICCG and the JCG solvers (**EQSLV** command). You must issue the **THOPT** command before the **EQSLV** command to use the QUASI option.

ANTYPE,,RESTART is not supported by **THOPT**,QUASI and **THOPT**, LINEAR options.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>Analysis Options

TIFF, *Kywrd*, *OPT*

Provides TIFF file Export for ANSYS Displays.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Kywrd

Specifies various TIFF file export options.

COMP

If *Kywrd* = COMP, then *OPT* controls data compression for the output file. If COMP = 0, then compression is off. If COMP = 1 (default), then compression is on.

ORIENT

If *Kywrd* = ORIENT, then *OPT* will determine the orientation of the entire plot. *OPT* can be either Horizontal (default) or Vertical.

COLOR

If *Kywrd* = COLOR, then *OPT* will determine the color attribute of the saved file. *OPT* can be 0, 1, or 2, corresponding to Black and White, Grayscale, and Color (default), respectively.

TMOD

If *Kywrd* = TMOD, then *OPT* will determine the text method. *OPT* can be either 1 or 0, corresponding to bitmap text (default) or line stroke text, respectively.

DEFAULT

If *Kywrd* = DEFAULT, then all of the default values, for all of the Kywrd parameters listed above, are active.

OPT

OPT can have the following names or values, depending on the value for *Kywrd* (see above).

1 or 0

If *Kywrd* = COMP, a value or 1 (on) or 0 (off) will control compression for the TIFF file.

Horizontal, Vertical

If *Kywrd* = ORIENT, the terms Horizontal or Vertical determine the orientation of the plot.

0, 1, 2

If *Kywrd* = COLOR, the numbers 0, 1, and 2 correspond to Black and White , Grayscale and Color, respectively.

1, 0

If *Kywrd* = TMOD, the values 1 and 0 determine whether bitmap (1) or stroke text (0) fonts will be used

Menu Paths

Utility Menu>PlotCtrls>HardCopy>ToFile

TIME, *TIME*

Sets the time for a load step.

SOLUTION: Load Step Options

MP ME ST DY <> PR EM <> <> PP ED

TIME
Time at the end of the load step.

Command Default

Previous *TIME* + 1.0 (at each load step), i.e., also corresponds to the load step number.

Notes

Associates the boundary conditions at the end of the load step with a particular *TIME* value. *TIME* must be a positive, nonzero, monotonically increasing quantity that "tracks" the input history. Typically, for the first load step *TIME* defaults to 1. However, for the first load step of a reduced transient analysis (**ANTYPE**,TRANS *and* **TRNOPT**,REDUC) or mode superposition transient analysis (**ANTYPE**,TRANS *and* **TRNOPT**,MSUP), the **TIME** command is ignored and a static solution is performed at *TIME* = 0. *TIME* is not used for the modal (**ANTYPE**,MODAL), harmonic response (**ANTYPE**,HARMIC), or substructure (**ANTYPE**,SUBSTR) analyses. Units of time should be consistent with those used elsewhere (for properties, creep equations, etc.).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substps
Main Menu>Solution>Analysis Type>Sol'n Controls>Basic
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substps
Main Menu>Solution>Time Controls>Solution Time

TIMERANGE, *TMIN*, *TMAX*

Specifies the time range for which data are to be stored.

POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

TMIN
Minimum time (defaults to first time (or frequency) point on the file).

TMAX
Maximum time (defaults to last time (or frequency) point on the file).

Command Default

Include all time (or frequency) points in the range.

Notes

Defines the time (or frequency) range for which data are to be read from the file and stored in memory. Use the **NSTORE** command to define the time increment.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Data

TIMINT, *Key*, *Lab*

Turns on transient effects.

SOLUTION: Dynamic Options

MP ME ST <> <> PR EM <> <> PP ED

Key

Transient effects key:

OFF
: No transient effects (static or steady-state).

ON
: Include transient (mass or inertia) effects.

Lab

Degree of freedom label:

ALL
: Apply this key to all appropriate labels (default).

STRUC
: Apply this key to structural DOFs.

THERM
: Apply this key to thermal DOFs.

ELECT
: Apply this key to electric DOFs.

MAG
: Apply this key to magnetic DOFs.

FLUID
: Apply this key to fluid DOFs.

Command Default

Include transient effects (ON) if **ANTYPE**,TRANS, exclude transient effects (OFF) if **ANTYPE**,STATIC.

Notes

Indicates whether this load step in a full transient analysis should use time integration, that is, whether it includes transient effects (e.g. structural inertia, thermal capacitance) or whether it is a static (steady-state) load step for the indicated DOFs. Transient initial conditions are introduced at the load step having *Key* = ON. Initial conditions

are then determined from the previous two substeps. Zero initial velocity and acceleration are assumed if no previous substeps exist. See the *ANSYS Structural Analysis Guide*, the *ANSYS Thermal Analysis Guide*, and the *ANSYS Low-Frequency Electromagnetic Analysis Guide* for details.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters
Main Menu>Solution>Analysis Type>Sol'n Controls>Transient
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters

---

TIMP, *ELEM*, *CHGBND*, *IMPLEVEL*

Improves the quality of tetrahedral elements that are not associated with a volume.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

ELEM

Identifies the tetrahedral elements to be improved. Valid values are ALL and P. If *ELEM* = ALL (default), improve all selected tetrahedral elements. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

CHGBND

Specifies whether to allow boundary modification. Boundary modification includes such things as changes in the connectivity of the element faces on the boundary and the addition of boundary nodes. (Also see the Notes section below for important usage information for *CHGBND*.)

0
Do not allow boundary modification.

1
Allow boundary modification (default).

IMPLEVEL

Identifies the level of improvement to be performed on the elements. (Improvement occurs primarily through the use of face swapping and node smoothing techniques.)

0
Perform the least amount of swapping/smoothing.

1
Perform an intermediate amount of swapping/smoothing.

2
Perform the greatest amount of swapping/smoothing.

3
Perform the greatest amount of swapping/smoothing, plus additional improvement techniques (default).

Notes

The **TIMP** command enables you to improve a given tetrahedral mesh by reducing the number of poorly-shaped tetrahedral elements (in particular, the number of sliver tetrahedral elements)--as well as the overall number of elements--in the mesh. It also improves the overall quality of the mesh.

TIMP is particularly useful for an imported tetrahedral mesh for which no geometry information is attached.

Regardless of the value of the *CHGBND* argument, boundary mid-nodes can be moved as long as you are not using p-method analysis. When *CHGBND* = 0 and you are using p-method analysis, boundary mid-nodes cannot be moved. (ANSYS issues an error message if it would be necessary to move boundary mid-nodes in order to generate valid quadratic elements.)

When loads or constraints have been placed on boundary nodes or mid-nodes, and boundary mid-nodes are later moved, ANSYS issues a warning message to let you know that it will not update the loads or constraints.

No boundary modification is performed if shell or beam elements are present in the mesh, even when *CHGBND* = 1.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Improve Tets>Detached Elems

---

TINTP, *GAMMA*, *ALPHA*, *DELTA*, *THETA*, *OSLM*, *TOL*, --, --, *AVSMOOTH*, *ALPHAF*, *ALPHAM*

Defines transient integration parameters.

SOLUTION: Dynamic Options

MP ME ST <> <> PR EM <> <> PP ED

GAMMA
Amplitude decay factor for 2nd order transient integration, e.g., structural dynamics (used only if *ALPHA*, *DELTA*, *ALPHAF*, and *ALPHAM* are blank). Defaults to 0.005.

ALPHA
2nd order transient integration parameter (used only if *GAMMA* is blank). Defaults to 0.2525.

DELTA
2nd order transient integration parameter (used only if *GAMMA* is blank). Defaults to 0.5050.

THETA
1st order transient (e.g., thermal transient) integration parameter. Defaults to 1.0.

OSLM
Specifies the oscillation limit criterion for automatic time stepping of 1st order transients (e.g., thermal transients). Defaults to 0.5 with a tolerance of *TOL*.

TOL
Tolerance applied to *OSLM*. Defaults to 0.0.

--, --
Unused fields.

AVSMOOTH
Smooth flag option:

0
Include smoothing of initial velocity (1st order system) or initial acceleration (2nd order system) (default).

1
Do not include smoothing.

ALPHAF
Interpolation factor in HHT algorithm for force and damping terms (used only if *GAMMA* is blank). Defaults to 0.005.

ALPHAM
Interpolation factor in HHT algorithm for inertial term (used only if *GAMMA* is blank). Defaults to 0.0.

Notes

Used to define the transient integration parameters. For more information on transient integration parameters, refer to the *ANSYS, Inc. Theory Reference*.

For structural transient analyses, you may choose between the Newmark and HHT time integration methods (see the **TRNOPT** command). In this case, if *GAMMA* is input and the integration parameters *ALPHA*, *DELTA*, *ALPHAF*, and *ALPHAM* are left blank, the program will calculate the integration parameters. Alternatively, you can input these integration parameters directly on this command. However, for the unconditional stability and second order accuracy of the time integration, these parameters should satisfy a specific relationship, as described in Section 17.2.2: Description of Structural and Other Second Order Systems of the *ANSYS, Inc. Theory Reference*.

In a transient piezoelectric analysis, required input for this command is *ALPHA* = 0.25, *DELTA* = 0.5, and *THETA* = 0.5. For a coupled electromagnetic-circuit transient analysis, use *THETA* = 1.0, the default value, to specify the backward Euler method.

The default values given for this command assume **SOLCONTROL**,ON (the default). See the description of **SOLCONTROL** for a complete listing of the defaults set by **SOLCONTROL**,ON and **SOLCONTROL**,OFF.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters
Main Menu>Solution>Analysis Type>Sol'n Controls>Transient
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters

/TITLE, *Title*

Defines a main title.

DATABASE: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Title
Input up to 72 alphanumeric characters. Parameter substitution may be forced within the title by enclosing the parameter name or parametric expression within percent (%) signs.

Notes

The title is carried through the printout and written on various files. The title written to a file is the title defined at that time. Special characters may be used within the title text. Subtitles may also be defined [**/STITLE**].

This command is valid in any processor.

Menu Paths

Utility Menu>File>Change Title

/TLABEL, *XLOC*, *YLOC*, *Text*

Creates annotation text (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

XLOC
Text *X* starting location (-1.0 < *X* < 1.6).

YLOC
Text Y starting location (-1.0 < *Y* < 1.0).

Text
Text string (60 characters maximum). Parameter substitution may be forced within the text by enclosing the parameter name or parametric expression within percent (%) signs.

Notes

Defines annotation text to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

All text is shown on subsequent displays unless the annotation is turned off or deleted. Use the **/TSPEC** command to set the attributes of the text.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

TOCOMP, *Refname*, *Type*, *NUMLC*, *LCARR*

Defines single or multiple compliance as the topological optimization function.

OPTIMIZATION: Specifications

MP ME ST <> <> PR <> <> <> <> ED

Refname
Reference name (8 character string). **TOCOMP**,*Refname* with the other fields blank deletes the defined compliance function.

Type
Sets the type of compliance function. Valid types are:

SINGLE
Sets the topological optimization function for *Refname* to single compliance. *NumLC* is the load case number.

MULTIPLE
Sets the topological optimization function to weighted summation of individual compliances. *NumLC* is the number of load cases considered

NUMLC
For *Type* = Single, *NumLC* is the load case identifier. For *Type* = Multiple, *NumLC* is the total number of load cases considered.

LCARR
Used when *Type* = Multiple. The array (of dimension *NumLC*) defining the load case weights. Array name must be enclosed in % signs: %WEIGHT%.

Notes

The function defined using **TOCOMP** is used as an objective or constraint for the topological optimization, as defined further in **TOVAR**. When used as the objective, the optimization distributes the material so that the compliance measure is minimized (stiffness is maximized).

Menu Paths

Main Menu>Topological Opt>Set Up>Advanced Opt>Topo Function
Main Menu>Topological Opt>Set Up>Basic Opt

TODEF, *ACCUR*

Defines parameters for and initializes topological optimization.

OPTIMIZATION: Specifications

MP ME ST <> <> PR <> <> <> <> ED

ACCUR
Accuracy used for termination and convergence checking. Default = 0.0005.

Notes

Initializes the topological optimization problem and defines the accuracy for the solution. Issue this command before **TOLOOP**.

Menu Paths

Main Menu>Topological Opt>Run

TOEXE

Executes one topological optimization iteration.

OPTIMIZATION: Run

MP ME ST <> <> PR <> <> <> <> ED

Notes

Runs one topological optimization iteration, leading to the prediction of a new shape, defined by means of element densities. Before issuing **TOEXE**, you must perform a static or modal analysis solution depending on the type of objective and constraints you defined. **TOEXE** performs a convergence test based on relative objective and element density change with an accuracy as specified by **TODEF**.

Menu Paths

This command cannot be accessed from a menu.

TOFFST, *VALUE*

Specifies the temperature offset from absolute zero to zero.

SOLUTION: Analysis Options

MP ME ST <> <> PR <> <> FL PP ED

VALUE

Degrees between absolute zero and zero of temperature system used (should be positive).

Notes

Specifies the difference (in degrees) between absolute zero and the zero of the temperature system used. Absolute temperature values are required in evaluating certain expressions, such as for creep, swelling, radiation heat transfer, MASS71, etc. (The offset temperature is not used in evaluating emissivity.) Examples are 460° for the Fahrenheit system and 273° for the Centigrade system. The offset temperature is internally included in the element calculations and does not affect the temperature input or output. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Temperature Units
Main Menu>Preprocessor>Material Props>Temperature Units
Main Menu>Preprocessor>Radiation Opts>Solution Opt
Main Menu>Radiation Opt>Radiosity Meth>Solution Opt
Main Menu>Solution>Analysis Type>Analysis Options
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Temperature Units
Main Menu>Solution>Radiation Opts>Solution Opt

TOFREQ, *Refname*, *Type*, *Nfreq*, *Frqarr*, *Targval*

Defines single or mean frequency formulation as the topological optimization function.

OPTIMIZATION: Specifications

MP ME ST <> <> PR <> <> <> <> ED

Refname
Reference name (8 character string). **TOFREQ**,*Refname* with other fields blank deletes the defined compliance function.

Type
Defines type of frequency topological optimization function. Valid types are:

SINGLE
Defines single frequency as the topological optimization function.

WEIGHTED
Defines a weighted summation of individual frequencies as the topological optimization function.

RECIPROCAL
Uses a reciprocal formulation as the topological optimization function.

EUCLIDEAN
Uses a Euclidean Norm formulation as the topological optimization function.

Nfreq
For *Type* = Single, *Nfreq* is the frequency identifier. For *Type* = WEIGHTED, RECIPROCAL, or EUCLIDEAN, *Nfreq* is the total number of frequencies considered.

Frqarr
For *Type* = WEIGHTED or RECIPROCAL, *Frqarr* is the array of dimension *Nfreq* that defines the weights for mean frequency formulation. For *Type* = EUCLIDEAN, *Frqarr* is the array (of dimension *Nfreq*) defining the target values for mean frequency formulation. The array name must be enclosed in % signs: %ARRAY%.

Targval
For *Type* = RECIPROCAL, *Targval* specifies the target value for mean frequency formulation.

Type	Nfreq	Frqarr	Targval
SINGLE	Frequency identifier	N/A	N/A
WEIGHTED	Total number of frequencies	Defines weights	N/A
RECIPROCAL	Total number of frequencies	Defines weights	Target value
EUCLIDEAN	Total number of frequencies	Defines target values	N/A

Notes

Single, when the frequency function is set as the objective (**TOVAR**), distributes material so that the single frequency specified is maximized.

Weighted (available only as an objective function) distributes material so that the specified mean frequency formulation is maximized.

Reciprocal (available only as an objective function) is a mean frequency formulation that defines a smoother function than Single. It is best used when two modes whose eigenfrequencies occur in the given mean formulation exchange their orders during optimization. The eigenfrequency that is closest to the target frequency (set with *Targval*) experiences the largest increase and is maximized.

Euclidean Norm Formulation (available only as an objective function) is used to shift single or multiple eigenfrequencies up. The mean eigenfrequency is utilized to smooth the optimization process. This must be used as the objective where the optimization procedure distributes material so that the mean frequency is minimized.

If the optimization problem does not converge, try specifying a different type of frequency formulation. Each formulation uses different methods to calculate the frequency optimization, so depending on your specific problem, one might work better than another. See the *ANSYS, Inc. Theory Reference* for more information on these formulations.

Menu Paths

Main Menu>Topological Opt>Set Up>Advanced Opt>Topo Function

TOGRAPH, *Type*, *Refname*

Plots iteration solution of topological optimization.

OPTIMIZATION: Display

MP ME ST <> <> PR <> <> <> <> ED

Type

OBJ
: Plot objective history versus iteration number.

CON
: Plot constraint history versus iteration number.

Refname
: Used if multiple constraints were defined. Defaults to the first constraint name.

Notes

Plots all iterations of the topological optimization solution, using either objective or constraint history.

Menu Paths

Main Menu>Topological Opt>Graph History

TOLIST

Lists all topological optimization functions currently defined.

OPTIMIZATION: Display

MP ME ST <> <> PR <> <> <> <> ED

Notes

Provides a list of all defined topological functions, in the order they were defined.

Menu Paths

Main Menu>Topological Opt>Set Up>Advanced Opt>List Functions

TOLOOP, *NITER*, *PLOT*

Execute several topological optimization iterations.

OPTIMIZATION: Run

MP ME ST <> <> PR <> <> <> <> ED

NITER
Number of iterations to be performed (maximum of 100). Default is 1.

PLOT
Sets display of topological densities for each iteration:

0
Do not display the results of each iteration.

1
Display the results of each iteration.

Command Default

TOLOOP,1,0.

Notes

Invokes a macro to solve, postprocess, and plot each iteration. For compliance cases, you must write at least one load step (using **LSWRITE**) before issuing this command. **LSSOLVE** is used for static analyses if there are multiple load steps. The Block Lanczos eigensolver is used for modal analyses. **PLNSOL**,TOPO and **TOEXE** are used for each iteration. The macro terminates when either the number of iterations or the convergence criteria is met, as specified in **TODEF**,*ACCUR*.

Menu Paths

Main Menu>Topological Opt>Run

TOPLOT, *AVRG*

Plot current topological density distribution.

OPTIMIZATION: Display

MP ME ST <> <> PR <> <> <> <> ED

AVRG

Sets **TOPLOT** to show smoothed nodal solution or non-smoothed element solution.

0

Plot smoothed nodal solution. Default.

1

Plot non-smoothed element solution.

Notes

Invokes a macro to postprocess the current element pseudo densities for topological optimization. **TOPLOT**,0 uses **PLNSOL**,TOPO, and **TOPLOT**,1 uses **PLESOL**,TOPO to plot the current density distribution predicted by the topological optimization.

Menu Paths

Main Menu>Topological Opt>Plot Dens Unavg
Main Menu>Topological Opt>Plot Densities

TOPRINT, *Type*, *Refname*

Print iteration solution history of topological optimization.

OPTIMIZATION: Display

MP ME ST <> <> PR <> <> <> <> ED

Type

OBJ

Print objective history versus iteration number.

CON

Print constraint history versus iteration number.

Refname

Used if multiple constraints were defined. Defaults to the first constraint name.

Notes

Prints one iteration of the topological optimization solution, using either objective or constraint history.

Menu Paths

Main Menu>Topological Opt>Print History

TORQ2D

Calculates torque on a body in a magnetic field.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

Notes

TORQ2D invokes an ANSYS macro which calculates mechanical torque on a body in a magnetic field. The body must be completely surrounded by air (symmetry permitted), and a closed path [**PATH**] passing through the air elements surrounding the body must be available. A counterclockwise ordering of nodes on the **PPATH** command will give the correct sign on the torque result. The macro is valid for 2-D planar analysis.

The calculated torque is stored in the parameter TORQUE. A node plot showing the path is produced in interactive mode. The torque is calculated using a Maxwell stress tensor approach. Path operations are used for the calculation, and all path items are cleared upon completion. See the **TORQC2D** command for torque calculation based on a circular path.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>Torque

TORQC2D, *RAD*, *NUMN*, *LCSYS*

Calculates torque on a body in a magnetic field based on a circular path.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

RAD
Radius of the circular path. The nodes for the path are created at this radius.

NUMN
Number of nodes to be created for the circular path. The greater the number of nodes, the higher the accuracy of the torque evaluation. Defaults to 18.

LCSYS
(Optional) Local coordinate system number to be used for defining the circular arc of nodes and the path. Defaults to 99. (If a local system numbered 99 already exists, it will be overwritten by this default.)

Notes

TORQC2D invokes an ANSYS macro which calculates the mechanical torque on a body using a circular path. It is used for a circular or cylindrical body such as a rotor in an electric machine. The body must be centered about the global origin and must be surrounded by air elements. The air elements surrounding the path at radius *RAD* must be selected, and elements with a high-permeability material should be unselected prior to using the macro. The macro is valid for 2-D planar analyses only. For a harmonic analysis, the macro calculates the time-average torque. Radial symmetry models are allowed, i.e., the model need not be a full 360 ° model.

The calculated torque is stored in the parameter TORQUE. If the model is not a full 360° model, TORQUE should be multiplied by the appropriate factor (such as 4.0 for a 90 ° sector) to obtain the total torque. A node plot showing the path is produced in interactive mode.

The torque is calculated via a circular path integral of the Maxwell stress tensor. The circular path and the nodes for the path are created by the macro at the specified radius *RAD*. Path operations are used for the calculation, and all path items are cleared upon completion. See the **TORQ2D** command for torque calculation based on an arbitrary, non-circular path.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>Circular Torq

---

TORQSUM, *Cnam1*, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*, *Cnam8*, *Cnam9*

Summarizes electromagnetic torque calculations on element components.

POST1: Magnetics Calculations

MP ME ST <> <> <> EM <> <> PP ED

Cnam1, *Cnam2*, *Cnam3*, *Cnam4*, *Cnam5*, *Cnam6*, *Cnam7*, *Cnam8*, *Cnam9*

Names of existing element components for which Maxwell or virtual work boundary conditions were applied in the preprocessor. Must be enclosed in single quotes (e.g., 'CNAM1') when the command typed in the command input box.

Notes

TORQSUM invokes an ANSYS macro that summarizes the Maxwell and virtual work torque values. The element components must have had appropriate Maxwell or virtual work boundary conditions established in the preprocessor prior to solution in order to retrieve torques (see the **FMAGBC** command). The torque values are stored on a per-element basis for the adjacent air layer elements surrounding the components and are retrieved and summed by the macro. For a harmonic analysis, the calculated torque represents a time-average value.

TORQSUM is valid only for 2-D planar analysis.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Component Based>Torque

---

TORUS, *RAD1*, *RAD2*, *RAD3*, *THETA1*, *THETA2*

Creates a toroidal volume.

PREP7: Primitives

MP ME ST DY <> PR EM EH FL PP ED

RAD1, *RAD2*, *RAD3*

Three values that define the radii of the torus. You can specify the radii in any order. The smallest of the values is the inner minor radius, the intermediate value is the outer minor radius, and the largest value is the major radius. (There is one exception regarding the order of the radii values--if you want to create a solid torus, specify zero or blank for the inner minor radius, in which case the zero or blank *must* occupy either the *RAD1* or *RAD2* position.) At least two of the values that you specify must be positive values; they will be used to define the outer minor radius and the major radius. See the diagram in the Notes section for a view of a toroidal sector showing all radii.

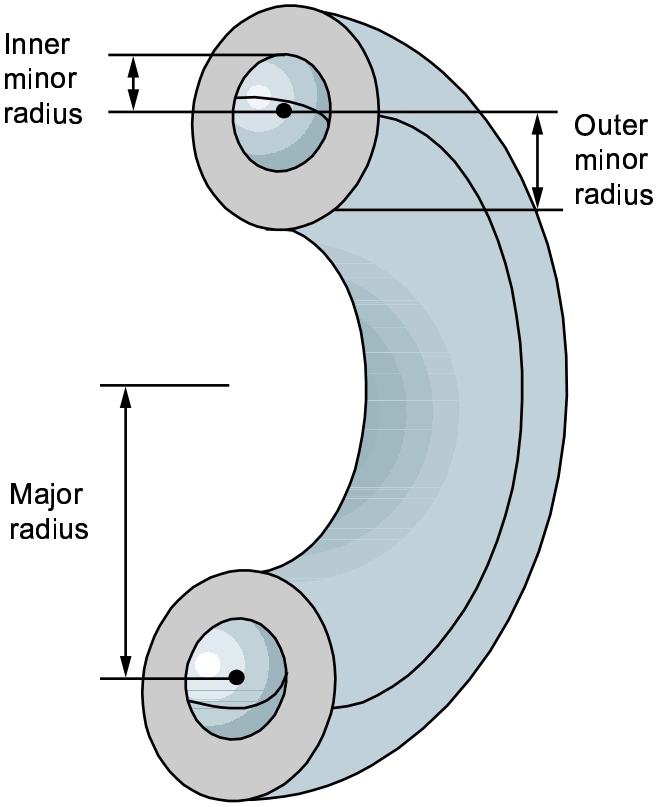
THETA1, *THETA2*

Starting and ending angles (either order) of the torus. Used for creating a toroidal sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0° and the ending angle defaults to 360°.

Notes

Defines a toroidal volume centered about the working plane origin. A solid torus of 360° will be defined with four areas, each area spanning 180° around the major and minor circumference.

To create the toroidal sector shown below, the command **TORUS**,5,1,2,0,180 was issued. Since "1" was the smallest radii value specified, it defined the inner minor radius; since "2" was the intermediate radii value specified, it defined the outer minor radius; and since "5" was the largest radii value specified, it defined the major radius. The values "0" and "180" defined the starting and ending angles of the torus.



Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Torus

TOSTAT

Displays topological optimization status and results information.

OPTIMIZATION: Display

MP ME ST <> <> PR <> <> <> <> ED

Notes

Displays information for defined functions from **TOVAR**, **TODEF**, and **TOTYPE**, and results information such as final volume, number of iterations, and convergence status.

Menu Paths

Main Menu>Topological Opt>Set Up>Advanced Opt>Status
Main Menu>Topological Opt>Status

TOTAL, *NTOT*, *NRMDF*

Specifies automatic MDOF generation.

SOLUTION: Master DOF

MP ME ST <> <> PR <> <> <> PP ED

NTOT
Total number of master degrees of freedom to be used in the analysis, including specified (NS, see below) master degrees of freedom. *NTOT* must be greater than NS if any automatic generation is to be done.

NRMDF
Rotational masters key:

0
Include all degrees of freedom in automatic master selection.

1
Exclude rotational degrees of freedom (and VOLT degrees of freedom in a piezoelectric analysis) from automatic selection.

Command Default

Do not use any automatically generated MDOF.

Notes

Specifies automatic master degree of freedom (MDOF) generation. The limit on the number of MDOF is equal to the maximum in-memory wavefront size (see the *ANSYS Basic Analysis Guide*). If NS is defined as the number of master degrees of freedom specified with the **M** or **MGEN** command, *NTOT*-NS additional master degrees of freedom will be automatically generated *during the solution phase* if **TOTAL** is used. NS may be zero, i.e., all master degrees or freedom can be automatically generated. After the solution phase, generated masters become specified masters (NS = *NTOT*) so that they may be listed, displayed, modified, etc. The **TOTAL** command is ignored in subsequent solutions unless masters are deleted, such that NS<*NTOT*. If used in SOLUTION, this command is valid only within the first load step.

During the matrix triangulation (wavefront) operation, the first *NTOT* degrees of freedom are temporarily identified as masters and then are replaced as degrees of freedom with lower K/M ratios are found. Degrees of freedom matching the user specified set (if any) are permanently identified. The wavefront builds to *NTOT* and will have a minimum (and final) value of *NTOT*. The final set of automatic masters identified will be those corresponding to the lowest modes of the structure.

Constrained degrees of freedom are excluded from the automatic master selection. Constraints may be defined to prevent undesirable modes from being present (thus preventing the corresponding MDOF from being selected). For example, if symmetry constraints are imposed, degrees of freedom producing only symmetric modes will be selected. In-plane rotational degrees of freedom for shell elements lying in a global plane are automatically excluded. All rotational degrees of freedom can be excluded during the automatic selection if desired.

If automatic master selection is used in the reduced linear transient dynamic (**ANTYPE**,TRANS) analysis or the reduced harmonic response (**ANTYPE**,HARMIC) analysis, be sure to force the selection [**M**] of any degrees of freedom having nonzero displacement or force inputs. If automatic master selection is used in the superelement generation pass (**ANTYPE**,SUBSTR), be sure to force the selection of connection points to nonsuperelements.

Automatically selected masters are shown in the solution listing (and not in preprocessing listings) as follows:

- in the reduced eigenvector solution for modal (**ANTYPE**,MODAL).
- in the reduced displacement solution for harmonic response (**ANTYPE**,HARMIC).
- in the reduced displacement solution for linear transient dynamic (**ANTYPE**,TRANS).
- in the matrix or load vector printout for substructures (**ANTYPE**,SUBSTR).

In the substructure generation pass (**ANTYPE**,SUBSTR), a mass matrix must be available if the **TOTAL** option is to be used.

If you select the sparse solver as the equation solver, any **TOTAL** commands are ignored. The **TOTAL** command is only valid for the frontal solver.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Master DOFs>Program Selected
Main Menu>Solution>Master DOFs>Program Selected

TOTYPE, *Type*

Specifies solution method for topological optimization.

OPTIMIZATION: Specifications

MP ME ST <> <> PR <> <> <> <> ED

Type

Choose the solution method to use:

OC

Use Optimality Criteria (OC) approach. (Default)

SCP

Use Sequential Convex Programming (SCP) approach.

Notes

The OC approach is applicable to problems with only volume as the constraint (or "volume as the only constraint"). The SCP approach is applicable to all valid combinations of objectives and constraints. See **TOVAR** for a description of valid combinations.

Menu Paths

Main Menu>Topological Opt>Run

TOVAR, *Refname*, *Type*, *LOWER*, *UPPER*, *Boundtype*

Specifies the objective and constraints for the topological optimization problem.

OPTIMIZATION: Specifications

MP ME ST <> <> PR <> <> <> <> ED

Refname

Reference name (8 character string), previously defined with **TOCOMP** and **TOFREQ**, or VOLUME (reserved name; default).

Type

Valid types for this command are:

OBJ

Specifies the objective for the topological optimization problem. The *Refname*(s) specified must already be defined (**TOCOMP**,*Refname*, or **TOFREQ**) before defining your constraint(s). *LOWER*, *UPPER*, and *Boundtype* are not used if the specified *Refname* is the objective. To specify total volume as the objective function, use VOLUME as the *Refname*. For single or multiple compliance, see **TOCOMP**. For frequency formulations (single, weighted mean, reciprocal mean, or Euclidean normal), see **TOFREQ**.

CON

Specifies the constraint for the topological optimization problem. Requires a previously defined objective function (**TOVAR**,*Refname*,OBJ). *LOWER*, *UPPER*, and *Boundtype* must be specified as listed below.

DEL

Deletes the previously defined topological optimization objective or constraint named in *Refname*. *LOWER*, *UPPER*, and *Boundtype* are not used.

LOWER

Lower bound for the constraint (*Type* = CON).

UPPER

Upper bound for the constraint (*Type* = CON). Default is no defined upper bound.

Boundtype

For *Type* = CON, specifies whether the specified bounds are actual values, or indicate percentages.

PERCENT

Indicates that the values specified in *LOWER* and *UPPER* should be treated as percentages of the original value. The original value is 100%; only values less than 100 are valid. Depending on the type of constraint, this value indicates a decrease or increase of the initial values. For example, specify 50 to reduce the volume by 50%, or specify 30 to increase the compliance by 30%.

ACTUAL
Indicates that the values specified in *LOWER* and *UPPER* should be treated as actual values.

Command Default

TOVAR,VOLUME,OBJ

Notes

You must set the objective first, then set the constraint(s).

If a single or multiple compliance function (see **TOCOMP**) is specified as objective function, only the VOLUME function is allowed as a constraint.

If a single, weighted mean, reciprocal mean, or euclidean norm frequency (see **TOFREQ**) objective is selected, only the VOLUME function is allowed as a constraint.

If the VOLUME function is selected as the objective function, then only a single or multiple compliance constraint (see **TOCOMP**) is allowed as constraint function. In this case, a multiple single compliance constraint definition is also possible. LOWER is not applicable if *Type* = OBJ or DEL, and is not used if *Boundtype* = PERCENT.

UPPER is not applicable if *Type* = OBJ or DEL. If *Boundtype* = PERCENT, and *RefName* = VOLUME, UPPER specifies the percentage of volume reduction. If *Boundtype* = PERCENT, and *RefName* is a single or multiple compliance function, UPPER specifies the percentage of compliance increase.

If *Boundtype* = PERCENT, the valid range for *UPPER* is [5%, 95%].

Menu Paths

Main Menu>Topological Opt>Set Up>Advanced Opt>Topo Objective
Main Menu>Topological Opt>Set Up>Advanced Opt>TopoConstraint>By Percentage
Main Menu>Topological Opt>Set Up>Advanced Opt>TopoConstraint>By Value
Main Menu>Topological Opt>Set Up>Advanced Opt>TopoConstraint>Delete
Main Menu>Topological Opt>Set Up>Basic Opt

TRANS, *Fname*, *Ext*, --

Reformats File.GRPH for improved performance with plotters.

DISPLAY: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **TRAN33**.

Ext
Filename extension (8 character maximum).

--
Unused field

Notes

Reformats current ***Fname*.GRPH** data (based on color) for improved performance with pen plotters.

Menu Paths

It is part of the DISPLAY command.

TRANSFER, *KCNTO*, *INC*, *NODE1*, *NODE2*, *NINC*

Transfers a pattern of nodes to another coordinate system.

PREP7: Nodes

MP ME ST DY <> PR EM <> FL PP ED

KCNTO
Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system.

INC
Increment all nodes in the given pattern by *INC* to form the transferred node pattern.

NODE1, *NODE2*, *NINC*
Transfer nodes from pattern beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and the pattern is all selected nodes [**NSEL**]. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component may be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Notes

Transfers a pattern of nodes from one coordinate system to another. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. Coordinate values are interpreted in the active coordinate system and are transferred directly.

A model generated in one coordinate system may be transferred to another coordinate system. The user may define several coordinate systems (translated and rotated from each other), generate a model in one coordinate system, and then repeatedly transfer the model to other coordinate systems. The model may be generated in any type of coordinate system (Cartesian, cylindrical, etc.) and transferred to any other type of coordinate system. Coordinate values (X, Y, Z, or R, θ, Z, or etc.) of the model being transferred are interpreted in the *active* coordinate system type, regardless of how they were generated. Values are transferred directly and are interpreted according to the type of coordinate system being transferred to. For example, transferring from a Cartesian coordinate system to a cylindrical coordinate system (not recommended) would cause X = 2.0 and Y = 3.0 values to be directly interpreted as R = 2.0 and θ = 3.0 values, respectively.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Nodes

---

TREF, *TREF*

Defines the reference temperature for the thermal strain calculations.

SOLUTION: Load Step Options

MP ME ST DY <> PR <> <> <> PP ED

TREF

Reference temperature for thermal expansion.

Note — If the uniform temperature [**TUNIF**] is undefined, it is also set to this value.

Command Default

Reference temperature is 0.0 degrees.

Notes

Defines the reference temperature for the thermal strain calculations in structural analyses and explicit dynamic analyses. Thermal strains are given by α*(T-TREF), where α is the coefficient of thermal expansion (for more on this see the *ANSYS, Inc. Theory Reference*). You input the strain using ALPX, ALPY, ALPZ (the secant or mean coefficient value), or CTEX, CTEY, CTEZ (the instantaneous coefficient value), or the thermal strain value (THSX, THSY, THSZ). T is the element temperature. If α is temperature-dependent, TREF should be in the range of temperatures you define using the MPTEMP command.

Reference temperatures may also be input per material by using the label REFT on the material property [**MP**] command, such as **MP**,REFT,*MAT*,*C0*. Only a constant (non-temperature-dependent) value is allowed. The value input on the **TREF** command applies to all materials not having a specified material property definition.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Reference Temp
Main Menu>Preprocessor>Loads>Load Step Opts>Other>Reference Temp
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Reference Temp
Main Menu>Solution>Define Loads>Settings>Reference Temp
Main Menu>Solution>Load Step Opts>Other>Reference Temp
Main Menu>Solution>Loading Options>Reference Temp

/TRIAD, *Lab*

Shows the global XYZ coordinate triad on displays.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

Lab

Display triad as follows:

ORIG

Display triad at global origin (default).

OFF
Turn off triad display.

LBOT
Display triad in lower left screen corner.

RBOT
Display triad in lower right screen corner.

LTOP
Display triad in upper left screen corner.

RTOP
Display triad in upper right screen corner.

Notes

For efficiency, ANSYS 3-D graphics logic maintains a single data structure (segment), which includes the triad as a 3-D data object. If a 3-D device is involved (**/SHOW**,3D), and the ANSYS graphics are not being displayed as multi-plots, then the triad location is determined by the view settings for Window #1. A request for triad display anywhere except for the origin could yield an improper display in windows 2 through 5. The program displays the same segment in all windows. The view settings of each window constitute the only difference in the display in the active windows.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Window Controls>Reset Window Options
Utility Menu>PlotCtrls>Window Controls>Window Options

/TRLCY, *Lab*, *TLEVEL*, *N1*, *N2*, *NINC*

Specifies the level of translucency.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

Lab
Apply translucency level to the items specified by the following labels:

ELEM
Elements. Use *N1*, *N2*, *NINC* fields for element numbers.

AREA
Solid model areas. Use *N1*, *N2*, *NINC* fields for area numbers.

VOLU
Solid model volumes. Use *N1*, *N2*, *NINC* fields for volume numbers.

ISURF
Isosurfaces (surfaces of constant stress, etc., value). Translucency varies with result value, to a maximum of the specified translucency level.

CM
Component group. Use *N1* for component name, ignore *N2* and *NINC*.

CURVE
Filled areas under curves of line graphs. Use *N1*, *N2*, *NINC* fields for curve numbers.

ZCAP
If **/TYPE**,*WN*,ZCAP is the current display type, then **/TRLCY**,ZCAP,*TLEVEL* will display the model in window *WN* with the portion of the model in front of the section plane displayed at the translucency level *TLEVEL*.

ON, OFF
Sets the specified translucency display on or off. All other fields are ignored.

TLEVEL
Translucency level: 0.0 (opaque) to 1.0 (transparent).

N1, *N2*, *NINC*
Used only with labels as noted above. Apply translucency level to *Lab* items numbered *N1* to *N2* (defaults to *N1*) in steps of *NINC* (defaults to 1). If *N1* is blank or ALL, apply specified translucency level to entire selected range. If *Lab* is CM, use component name for *N1* and ignore *N2* and *NINC*. A value of *N1* = P allows you to graphically pick elements, areas, and volumes. You can then assign translucency levels to the entities via the picker. The *Lab* and *TLEVEL* fields are ignored when translucency is applied by picking.

Command Default

Zero translucency (opaque) level.

Notes

Specifies the level of translucency for various items. Issue **/TRLCY**,DEFA to reset the default (0) translucency levels. This command is valid only on selected 2-D and 3-D graphics devices; see in the *ANSYS Basic Analysis Guide* for more information on applying translucency.

For 2-D devices, ANSYS displays only the visible faces of the items being displayed. The information behind the facing planes is not displayed. Issuing the **/SHRINK** command will force the hardware to display information behind the translucent items.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Translucency

TRNOPT, *Method*, *MAXMODE*, *Dmpkey*, *MINMODE*, *MCout*, *TINTOPT*
Specifies transient analysis options.

SOLUTION: Dynamic Options

MP ME ST <> <> <> EM EH <> PP ED

Method
Solution method for the transient analysis:

FULL
Full method (default).

REDUC
Reduced method.

MSUP
Mode superposition method.

MAXMODE
Largest mode number to be used to calculate the response (for *Method* = MSUP). Defaults to the highest mode calculated in the preceding modal analysis.

Dmpkey
Damping option (for *Method* = REDUC):

DAMP
Include the effects of damping if present (default).

NODAMP
Ignore the effects of damping, even if present.

MINMODE
Smallest mode number to be used (for *Method* = MSUP). Defaults to 1.

MCout
Modal coordinates output key (valid only for the mode superposition method MSUP):

NO
No output of modal coordinates (default).

YES
Output modal coordinates to the text file **jobname.MCF**.

TINTOPT
Time integration method for the transient analysis:

NMK or 0
Newmark algorithm (default).

HHT or 1
HHT algorithm (valid only for the full transient method).

Notes

Specifies transient analysis (**ANTYPE**,TRANS) options. If used in SOLUTION, this command is valid only within the first load step. Use the **TINTP** command to set transient integration parameters.

This command is also valid in PREP7.

Product Restrictions

TRNOPT is not a valid command in ANSYS Professional. In ANSYS Professional, *Method* is automatically set to FULL for pure thermal transients and is set to MSUP for all other transient analyses.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options
Main Menu>Preprocessor>Loads>Analysis Type>New Analysis

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient
Main Menu>Solution>Analysis Type>Analysis Options
Main Menu>Solution>Analysis Type>New Analysis
Main Menu>Solution>Analysis Type>Sol'n Controls>Transient

TRPDEL, *NTRP1*, *NTRP2*, *TRPINC*

Deletes particle flow or charged particle trace points.

POST1: Trace Points

MP ME ST DY <> <> <> <> FL PP ED

NTRP1, *NTRP2*, *TRPINC*

Delete points from *NTRP1* to *NTRP2* (defaults to *NTRP1*) in steps of *TRPINC* (defaults to 1). If *NTRP1* = ALL, *NTRP2* and *TRPINC* are ignored and all trace points are deleted. If *NTRP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Deletes particle flow or charged particle trace points defined with the **TRPOIN** command.

Menu Paths

Main Menu>General Postproc>Plot Results>Dele Trace Pt

TRPLIS, *NTRP1*, *NTRP2*, *TRPINC*, *Opt*

Lists the particle flow or charged particle trace points.

POST1: Trace Points

MP ME ST DY <> <> <> <> FL PP ED

NTRP1, *NTRP2*, *TRPINC*

List points from *NTRP1* to *NTRP2* (defaults to *NTRP1*) in steps of *TRPINC* (defaults to 1). If *NTRP1* = ALL, *NTRP2* and *TRPINC* are ignored and all trace points are listed. If *NTRP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Opt

Opt = LOC lists the trace point number location (X, Y, Z). Default.

Opt = PART lists the trace point number particle settings (velocity, charge, mass).

Notes

Lists the particle flow or charged particle trace points in the active display coordinate system [**DSYS**]. Trace points are defined with the **TRPOIN** command.

Menu Paths

Main Menu>General Postproc>Plot Results>List Trace Pt

TRPOIN, *X*, *Y*, *Z*, *VX*, *VY*, *VZ*, *CHRG*, *MASS*

Defines a point through which a particle flow or charged particle trace will travel.

POST1: Trace Points

MP ME ST DY <> <> <> <> FL PP ED

X, *Y*, *Z*
Coordinate location of the trace point (in the active coordinate system). If *X* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

VX, *VY*, *VZ*
Particle velocities in the X, Y and Z directions (in the active coordinate system).

CHRG
Particle charge.

MASS
Particle mass.

Notes

Defines a point through which a particle flow or charged particle trace [**PLTRAC**] will travel. Multiple points (50 maximum) may be defined which will result in multiple flow traces. Use **TRPLIS** to list the currently defined trace points and **TRPDEL** to delete trace points.

The VX, VY, VZ, CHRG, and MASS arguments only apply to charged particles.

Menu Paths

Main Menu>General Postproc>Plot Results>Defi Trace Pt

TRTIME, *TIME*, *SPACING*, *OFFSET*, *SIZE*, *LENGTH*

Defines the options used for the PLTRAC (particle flow or charged particle trace) command.

POST1: Animation

MP ME ST DY <> PR EM <> FL PP ED

TIME
Total Trace Time (seconds) (defaults to 0, which is the full flow trace).

SPACING
Particle spacing in seconds (defaults to 0).

OFFSET
Particle offset in seconds (defaults to 0). It is used internally in the **ANFLOW** macro to produce an animation of particle flow in a flowing fluid or charged particle motion in an electric or magnetic field.

SIZE
Particle size (defaults to 0, which is a line).

LENGTH
Particle length fraction (defaults to .1).

Command Default

Full particle flow or charged particle trace.

Notes

The **TRTIME** command varies the type of **PLTRAC** display produced. Particle flow or charged particle traces follow a particle's path in the forward and backward direction of travel. The DOF selected determines the color of the particle trace. *SPACING* defines the particle spacing in seconds from adjacent particles in the stream line. *OFFSET* defines the offset in seconds from the spacing set by the *SPACING* argument.

LENGTH defines the particle length fraction. The default value (.1), means the particle occupies 10% of the flow region, and the other 90% is a color-coded line.

SIZE sets the radius of the particle. Use *SPACING*, *OFFSET* and *LENGTH* only when *SIZE* is nonzero (i.e., the particle is bigger than the line).

Menu Paths

Main Menu>General Postproc>Plot Results>Time Interval

TSHAP, *Shape*

Defines simple 2-D and 3-D geometric surfaces for target segment elements.

PREP7: Elements

MP ME ST <> <> PR EM <> <> PP ED

Shape

Specifies the geometric shapes for target segment elements TARGE169 and TARGE170.

LINE
: Straight line (2-D) (Default for 2-D)

PARA
: Parabola (2-D)

ARC
: Clockwise arc (2-D)

CARC
: Counterclockwise arc (2-D)

CIRC
: Complete circle (2-D)

TRIA
: Three-node triangle (3-D) (Default for 3-D)

TRI6
: Six-node triangle (3-D)

QUAD
: Four-node quadrilateral (3-D)

QUA8
Eight-node quadrilateral (3-D)

CYLI
Cylinder (3-D)

CONE
Cone (3-D)

SPHE
Sphere (3-D)

PILO
Pilot node (2-D, 3-D)

Notes

Use this command to generate the rigid target surface for surface-to-surface contact (TARGE169, CONTA171, CONTA172 (2-D) and TARGE170, CONTA173, and CONTA174 (3-D)). Once you issue **TSHAP**, all subsequent elements generated via this command will have the same shape, until you issue **TSHAP** again with a different *Shape*.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

---

/TSPEC, *TCOLOR*, *TSIZE*, *TXTHIC*, *PANGLE*, *IANGLE*

Creates annotation text attributes (GUI).

GRAPHICS: Annotation

MP ME ST DY <> PR EM <> FL PP ED

TCOLOR
Text color (0 ≤ *TCOLOR* ≤ 15):

0
Black.

1
Red-Magenta.

2
Magenta.

3
Blue-Magenta.

4
Blue.

5
Cyan-Blue.

6
Cyan.

7
Green-Cyan.

8
Green.

9
Yellow-Green.

10
Yellow.

11
Orange.

12
Red.

13
Dark Gray.

14
Light Gray.

15
White.

TSIZE
Text size factor.

TXTHIC
Text thickness key:

1
normal.

2
twice as thick.

3
three times as thick.

4
four times as thick.

PANGLE
Text path angle (0.0 < *angle* < 360.0).

IANGLE
Text italic angle (0.0 < *angle* < 45.0).

Notes

Defines annotation text attributes to control certain characteristics of the text created via the **/TLABEL** command. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if annotation is used. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

TSRES, *Array*

Defines an array of keytimes at which the time-stepping strategy changes.

SOLUTION: Load Step Options

MP ME ST DY <> PR EM <> FL PP ED

Array

Identifies an *N*x1x1 array parameter containing the keytimes at which the heat transfer time-stepping strategy changes (the time step is reset to the initial time step based on **DELTIM** or **NSUBST** settings). The array name must be enclosed by % signs (e.g., %*array*%). See ***DIM** for more information on array parameters.

Notes

Time values in the array parameter must be in ascending order and must not exceed the time at the end of the load step as defined on the **TIME** command. The time increment between time points in the array list must be larger than the initial time step defined on the **DELTIM** or **NSUBST** command. Time values must also fall between the beginning and ending time values of the load step. For multiple load step problems, you must either change the parameter values to fall between the beginning and ending time values of the load step or reissue the command with a new array parameter. To clear the array parameter specification, issue **TSRES**,ERASE. Results can be output at the requested time points if the array or time values in the array are also specified in the **OUTRES** command using *FREQ*=%*array*%. Use this command to reset the time-stepping strategy within a load step. You may need to reset the time-stepping strategy when using tabular time-varying boundary conditions.

See Steady-State Thermal Analysis of the *ANSYS Thermal Analysis Guide* for more information on applying boundary conditions via tabular input. See Transient Thermal Analysis of the *ANSYS Thermal Analysis Guide* for more information on defining keytimes.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substps
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substps

---

TUNIF, *TEMP*

Assigns a uniform temperature to all nodes.

SOLUTION: FE Body Loads

MP ME ST DY <> PR <> <> <> PP ED

TEMP

Uniform temperature assigned to the nodes. If *TEMP* is blank, the uniform temperature is set to zero.

Command Default

Set uniform temperature to the reference temperature ([**TREF**] but not **MP**,REFT).

Notes

In a transient or nonlinear thermal analysis, the uniform temperature is used during the first iteration of a solution as follows:

- as the starting nodal temperature (except where temperatures are explicitly specified [**D**, **DK**]),
- to evaluate temperature-dependent material properties.

In a structural analysis or explicit dynamic analysis, the uniform temperature is used as the *default* temperature for thermal strain calculations and material property evaluation (except where body load temperatures are specified [**BF**, **BFE**, **BFK**, **LDREAD**]). In other scalar field analyses, the uniform temperature is used for material property evaluation.

When the **TUNIF** command is used in an explicit dynamic analysis, you cannot use the **EDLOAD**,,TEMP command to apply temperature loading. Furthermore, temperature loading defined by **TUNIF** cannot be listed or deleted by the **EDLOAD** command.

TUNIF is a convenient form of the more general **BFUNIF** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>Uniform Temp
Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>Uniform Temp
Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Temperature>Uniform Temp
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>Uniform Temp
Main Menu>Preprocessor>Loads>Define Loads>Settings>Uniform Temp
Main Menu>Preprocessor>LS-DYNA Options>Loading Options>Uniform Temp
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>Uniform Temp
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>Uniform Temp
Main Menu>Solution>Define Loads>Apply>Structural>Temperature>Uniform Temp
Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>Uniform Temp
Main Menu>Solution>Define Loads>Settings>Uniform Temp
Main Menu>Solution>Loading Options>Uniform Temp

TVAR, *KEY*

Changes time to the cumulative iteration number.

POST26: Controls

MP ME ST DY <> PR EM <> FL PP ED

KEY

Time key:

0
Time is used for the variable *TIME*.

1
NCUMIT is used for the variable *TIME*.

Command Default

TIME is the variable *TIME*.

Notes

Changes the meaning of the time variable to the cumulative iteration number (NCUMIT) variable. Data can be read from the file, printed, and displayed as a function of NCUMIT rather than time. All POST26 descriptions applying to TIME then apply to NCUMIT.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Data

/TXTRE, *Lab*, *NUM*, *N1*, *N2*, *NINC*

Controls application of texture to selected items.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

Lab
You can apply texture according to the following labels:

ELEM
Apply texture to elements *N1* through *N2* in steps of *NINC*.

AREA
Apply texture to areas *N1* through *N2* in steps of *NINC*.

VOLU
Apply texture to volumes *N1* through *N2* in steps of *NINC*.

CM
Apply texture to the component named in *N1*. *N2* and *NINC* are ignored.

ON, OFF
Sets the specified texture display on or off. All other fields are ignored.

File
If Lab = File, the command format is **/TXTRE**, File, *Key_Index*, *Fname*, *Fext*, --, *Format* (This variant of the command is applicable to 2-D drivers).

Key_Index
The texture index associated with the file. If the number fifty-one (51) is used, the imported bitmap will be used as the window's logo.

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Fext
Filename extension (8 character maximum).

--
Unused field

Format
The file format. If *Format* = 0, the file is a pixmap (UNIX) or Bitmap (PC). The file cannot contain a compressed image, and the PC file must be 8 or 24 bit BI_RGB format. If *Format* = 1 or JPEG, then the file is in JPEG (Joint Photographic Experts Group) format. If *Format* = 2 or PNG, then the file is in PNG (Portable Network Graphics) format.

NUM
Select the texture index number from the following list:

0
No Texturing

1
Aluminum

2
Aluminum, Brushed

3
Steel With Bumps

4
Steel, Embossed

5
Iron

6
Steel, Pattern

7
Steel, Riveted

8
Steel, Scratched

9
Tin

10
Metal

11
Steel, Etched

12
Metal, Hot

13
Iron, Grainy

14
Metal, Rusty

15
Brick

16
Block

17
Wood

18
Wood, Light

19
Wood, Walnut

20
Plastic, Hard Blue

21
Plastic, Light Blue

22
Plastic, Hard Red

31
Gold

32
Brass

33
Silver

34
Plastic, Black

35
Plastic, Ivory

36
Plastic, Blue

37
Plastic, Red

38
Plastic, Yellow

39
Plastic, Green

40
Plastic, Brown

N1, *N2*, *NINC*
Apply texture to *Lab* items numbered *N1* through *N2* in steps of *NINC* (defaults to 1). If *Lab* = CM, then N1 is used to for the component name and *N2* and *NINC* are ignored. If *Lab* = ELEM, AREA, or VOLU and *N1* = blank or ALL, then the specified texture will be applied to all entities of type *Lab*. If *N1* = P, then graphical picking is enabled.

Command Default

No texture (**/TXTRE**,DEFA)

Notes

This command is available for 3-D Open GL devices. 2-D devices are supported **only** for the *Lab* = *File* variation of the command, allowing imported bitmaps to be used for texturing and annotation. Textures can affect the speed of many of your display operations. You can increase the speed by temporarily turning the textures off (**Utility Menu> PlotCtrls> Style> Texturing(3D)> Display Texturing**). This menu selection toggles your textures on and off. When textures are toggled off, all of the texture information is retained and reapplied when texturing is toggled back on.

For some displays, the texture will appear distorted because of a technique used to enhance 3-D displays (**/DV3D**,TRIS,1). Disabling this function (**/DV3D**,TRIS,0) will improve the quality of some texture displays. Disabling the TRIS option of the **/DV3D** command will slow down 3-D displays significantly. Be sure to reapply the TRIS option after you obtain a satisfactory output.

Specifying **/TXTRE**,DEFA removes all texturing.

Menu Paths

Utility Menu>PlotCtrls>Style>Texturing(3D)

/TYPE, *WN*, *Type*

Defines the type of display.

GRAPHICS: Style

MP ME ST DY <> PR EM <> FL PP ED

WN

Window number (or ALL) to which command applies (defaults to 1).

Type

Display type. Defaults to ZBUF for raster mode displays or BASIC for vector mode displays:

BASIC or 0
: Basic display (no hidden or section operations).

SECT or 1
: Section display (plane view). Use the **/CPLANE** command to define the cutting plane.

HIDC or 2
: Centroid hidden display (based on item centroid sort).

HIDD or 3
: Face hidden display (based on face centroid sort).

HIDP or 4
: Precise hidden display (like HIDD but with more precise checking). Because all facets are sorted, this mode can be extremely slow, especially for large models.

CAP or 5
: Capped hidden display (same as combined SECT and HIDD with model in front of section plane removed).

ZBUF or 6
Z-buffered display (like HIDD but using software Z-buffering).

ZCAP or 7
Capped Z-buffered display (same as combined SECT and ZBUF with model in front of section plane removed).

ZQSL or 8
QSLICE Z-buffered display (same as SECT but the edge lines of the remaining 3-D model are shown).

HQSL or 9
QSLICE precise hidden display (like ZQSL but using precise hidden).

Command Default

ZBUF for raster mode displays; BASIC for vector mode displays.

Notes

Defines the type of display, such as section display or hidden-line display. Use the **/DEVICE** command to specify either raster or vector mode.

The SECT, CAP, ZCAP, ZQSL, and HQSL options produce section displays. The section or "cutting" plane is specified on the **/CPLANE** command as either normal to the viewing vector at the focus point (default), or as the working plane.

When you use PowerGraphics, the section display options (Section, Slice, and Capped) use different averaging techniques for the interior and exterior results. Because of the different averaging schemes, anomalies may appear at the transition areas. In many cases, the automatically computed MIN and MAX values will differ from the full range of interior values. You can lessen the effect of these anomalies by issuing **AVRES**,,FULL (**Main Menu> General Post Proc> Options for Outp**). This command sets your legend's automatic contour interval range according to the minimum and maximum results found throughout the entire model.

With PowerGraphics active (**/GRAPHICS**,POWER), the averaging scheme for surface data with interior element data included (**AVRES**,,FULL) and multiple facets per edge (**/EFACET**,2 or **/EFACET**,4) will yield differing minimum and maximum contour values depending on the Z-Buffering options (**/TYPE**,,6 or **/TYPE**,,7). When the Section data is not included in the averaging schemes (**/TYPE**,,7), the resulting absolute value for the midside node is significantly smaller.

The HIDC, HIDD, HIDP, ZBUF, ZQSL, and HQSL options produce displays with "hidden" lines removed. Hidden lines are lines obscured from view by another element, area, etc. The choice of non-Z-buffered hidden-line procedure types is available only for raster mode [**/DEVICE**] displays. For vector mode displays, all non-Z-buffered "hidden-line" options use the same procedure (which is slightly different from the raster procedures). Both geometry and postprocessing displays may be of the hidden-line type. Interior stress contour lines within solid elements can also be removed as hidden lines, leaving only the stress contour lines and element outlines on the visible surfaces. Midside nodes of elements are ignored on postprocessing displays. Overlapping elements will not be displayed.

The ZBUF, ZCAP, and ZQSL options use a specific hidden-line technique called software Z-buffering. This technique allows a more accurate display of overlapping surfaces (common when using Boolean operations or **/ESHAPE** on element displays), and allows smooth shaded displays on all interactive graphics displays. Z-buffered displays can be performed faster than HIDP and CAP type displays for large models. See also the **/LIGHT**, **/SHADE**, and **/GFILE** commands for additional options when Z-buffering is used.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

TYPE, *ITYPE*

Sets the element type attribute pointer.

PREP7: Meshing
PREP7: Elements
MP ME ST DY <> PR EM <> FL PP ED

ITYPE
Assign this type number to the elements (defaults to 1).

Command Default

ITYPE = 1.

Notes

Activates an element type number to be assigned to subsequently defined elements. This number refers to the element type number (*ITYPE*) defined with the **ET** command. Type numbers may be displayed [**/PNUM**].

In some cases, ANSYS can proceed with a meshing operation even when no logical element type has been assigned via **TYPE** or ***X*ATT**,,,*TYPE*. For more information, see the discussion on setting element attributes in Meshing Your Solid Model in the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs
Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes
Main Menu>Preprocessor>Modeling>Operate>Extrude>Elem Ext Opts

---

TZAMESH, *Tvolu*, *SIZE*, *NDIV*

Meshes the areas of a volume to create Trefftz nodes.

PREP7: Trefftz Domain
MP ME ST <> <> <> EM <> <> PP ED

Tvolu
Component name for the Trefftz volume. The component name must be enclosed in single quotes in the **TZAMESH** command line.

SIZE
Edge length between Trefftz nodes on surface boundaries (i.e. lines). The areas of the component volume will be meshed with this specification to create the Trefftz nodes. If *SIZE* is zero (or blank), use *NDIV*.

NDIV
Number of element divisions along the surface boundary lines. Defaults to 2.

Notes

The command macro is used to create Trefftz nodes by meshing the surface areas of the selected volumes. The nodes of the meshed surface areas are grouped into the node component TZ_NOD. There is no solid model associativity with the Trefftz volume and the Trefftz nodes. The Trefftz nodes (component TZ_NOD) are used to create the Trefftz substructure (**TZEGEN** command).

Menu Paths

Main Menu>Preprocessor>Trefftz Domain>Mesh TZ Geometry

TZDELE

Deletes the Trefftz superelement, associated constraint equations and all supporting Trefftz files.

PREP7: Trefftz Domain

MP ME ST <> <> <> EM <> <> PP ED

Notes

Deletes the Trefftz superelement, associated constraint equations and all supporting Trefftz files previously generated with the **TZEGEN** command.

You should delete the Trefftz superelement if you are going to create new Trefftz nodes and generate a new Trefftz superelement.

Menu Paths

Main Menu>Preprocessor>Trefftz Domain>Superelement>Delete TZ

TZEGEN

Generates a Trefftz domain substructure and defines a Trefftz superelement for use in electrostatic analysis.

PREP7: Trefftz Domain

MP ME ST <> <> <> EM <> <> PP ED

Notes

The command generates a Trefftz domain substructure from the Trefftz node component (TZ_NOD) and the flagged infinite surfaces of the exterior finite element domain [**SF** command with the INF surface load label].

The Trefftz nodes should be uniformly scattered between the modeled components and the exterior of the finite element mesh. The nodes must not be attached to any underlying electrostatic finite elements. The nodes may be created automatically from a solid model volume using the **TZEGEN** command macro.

The substructure named **Jobname.sub** is created and automatically brought into the problem as a superelement (MATRIX50) using the next available element number. Also created is a set of constraint equations relating the Trefftz nodes to the surface nodes of the flagged exterior finite element domain. The **TZEGEN** command creates temporary files during the substructure generation. These include the following:

- **Jobname.TZN** - Trefftz nodes
- **Jobname.TZE** - Trefftz surface facets on the FE boundary
- **Jobname.TZX** - Surface nodes of the FE boundary
- **Jobname.TZM** - Trefftz material

The **TZEGEN** macro is only valid for 3-D electrostatics analysis. In addition, the bounding surface of the finite element domain must contain no symmetry planes.

Menu Paths

Main Menu>Preprocessor>Trefftz Domain>Superelement>Generate TZ

U Commands

/UDOC, *WIND*, *Class*, *Key*,

Determines position and content for the multi-legend options.

GRAPHICS: Labeling

MP ME ST DY <> PR EM <> FL PP ED

WIND

The window number to which the command applies. (defaults to 1)

Class

The type (and relative importance) of legend item being displayed:

CNTR

Contour legend. This legend item is controlled separately from the other legend items (see note below).

DATE

The items in the DATE class include the date and time, or the ANSYS graphical logo (**/PLOPTS**,LOGO,1). This item is shown by default in all plots.

GWIN

The items in the GWIN class include the entity acronyms that appear in the legend of a multiplot of entities (Nodes, Elements, Keypoints, Lines, Areas, Volumes). GWIN items are shown by default for all **GPLOT** displays.

TYPE

Items in the TYPE class include the plot type (e.g. ELEMENTS, MATERIALS, NODAL SOLUTIONS, etc.). TYPE items are shown by default in all plots.

TYP2

Items in the TYP2 class include supplementary type information, such as DMAX and SMAX for nodal solutions. TYP2 items are shown by default in all plots.

INUM

Items in the INUM class include the number labels generated by the **/PNUM** command. This class is displayed by default in all plots that contain **/PNUM** information.

BCDC

The items in the BCDC class include labels created by the **/PBC** command. This class is shown by default in all plots which contain **/PBC** information.

VECT

Items in the VECT class include labels created by the **PLVECT** command. This class is shown by default for all **PLVECT** plots.

SURF

The items in the SURF class include labels from the **/PSF** legend. This class is shown by default on all plots of surface boundary conditions.

BODY

Items from the BODY class include labels from the **/PBF** legend. This class is shown by default in all plots of body forces.

PSTA
: Items from the PSTA class include stress scaling statistics, such as the **/SSCALE** setting. This class is not shown as the default for any type of plot, and must be specifically referenced to display the included data.

VIEW
: The items in the VIEW class include view statistics. This class is not shown as the default for any type of plot, and must be specifically referenced to display the included data.

MISC
: The items in the MISC class include supplementary labels like /EXPANDED and Stress Section Cross Section. This class is not shown as the default for any type of plot, and must be specifically referenced to display the included data.

KEY
: Switch:

OFF or 0 --	Do not display the legend.
LEFT, RIGHT, TOP or BOTTOM --	If the value for *Class* is CNTR, these are the four acceptable values for the contour legend position.
LEFT or RIGHT --	If any value other than CNTR is used for *Class*, these are the two acceptable values for the text data.

Notes

The legend classes conform to the controls specified in the window options panel (**PlotCtrls> Window Controls> Window Options**). In many instances, the legend controls specified with the **/PLOPTS** command will take precedence and override **/UDOC** specifications. For instance:

/PLOPTS,LEG1,OFF will disable the TYPE, TYP2, INUM, and MISC classes, regardless of the **/UDOC** settings.

/PLOPTS,LEG2,OFF will disable the VIEW class, regardless of the **/UDOC** settings.

/PLOPTS,LEG3,OFF will disable the PSTA class, regardless of the **/UDOC** settings.

All items in a class are listed with the same X coordinate (except for contours). The contents of the text classes are dumped onto the display window from top to bottom, in order of class importance.

The font specification for text items that are included in the user-specified legends are controlled with the **/DEVICE** command (**PlotCtrls> Font Controls> Anno/Graph Font**).

The floating point values for the data presented in the legend(s) are controlled by the **/GFORMAT** command.

Menu Paths

Utility Menu>PlotCtrls>Style>MultiLegend Options>Contour Legend
Utility Menu>PlotCtrls>Style>MultiLegend Options>Text Legend

/UI, *Func*, *Type*, *Format*, *Screen*, *Color*, *Krev*, *Orient*, *Compress*, *Quality*
Activates specified GUI dialog boxes.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Func
Label identifying the dialog box to be activated:

HELP
Activates the online help system. Valid only in non-UI graphics mode (**/MENU**,GRPH).

VIEW
Activates the Pan, Zoom, Rotate dialog box

WPSE
Activates the Working Plane Settings dialog box.

WPVI
Activates the Offset Working Plane dialog box.

RESULT
Activates the Query Picking Menu for reviewing results.

QUERY
Activates the Query Picked Entities (preprocess) dialog box.

COPY
Activates the Hard Copy dialog box.

ANNO
Activates the 2D Annotation dialog box.

AN3D
Activates the 3D Annotation dialog box.

SELECT
Activates the Select Entities dialog box.

NSEL
Activates a picking menu to select nodes.

ESEL
Activates a picking menu to select elements.

KSEL
Activates a picking menu to select keypoints.

LSEL
Activates a picking menu to select lines.

ASEL
Activates a picking menu to select areas.

VSEL
Activates a picking menu to select volumes.

REFRESH
Refreshes the graphics window (non-UI mode only).

COLL
Controls the collapse of the ANSYS Main Menu when a **FINISH** command is issued. See *Type* below for a discussion of the arguments.

Type
Label identifying the type of select operation. Valid only for the following *Func* labels; NSEL, ESEL, KSEL, LSEL, ASEL, and VSEL:

S
Select a new set.

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

Label identifying the type of results data to be queried. Valid only for *Func* = RESULT:

NODE
Nodal solution data (h-elements only).

ELEMENT
Element solution data.

GRID
Subgrid solution data (p-elements only).

Label specifying the behavior of the ANSYS Main Menu after a **FINISH** command is issued. Note that this does not affect user interaction with the Main Menu. Valid only for *Func* = COLL:

YES, 1 or blank
Allows the Main Menu to collapse after **FINISH** command.

NO or 0
Prevents Main Menu collapse after **FINISH** command.

If *Func* = **COPY**, and *Type* = **SAVE,** command format is **/UI**,*Func*,*Type*,*Format*,*Screen*,*Color*,*Krev*,*Orient*,*Compress*,*Quality*. The remaining fields (after *Type*) identify the portion of the screen, the type of file, and the method of display that the portion of the screen is to be saved as.

Format

PSCR
Encapsulated Postscript File Format.

TIFF
Tagged Image File Format.

EPSI
Encapsulated Postscript with TIFF preview.

BMP
(PC only) Bitmap (Windows) file format.

WMF
(PC only) Windows Metafile format.

EMF
(PC only) Enhanced Metafile format.

JPEG
JPEG (Joint Photographic Experts Group) file format.

Screen

FULL
Saves the entire screen in the specified format.

GRAPH
Saves only the ANSYS Graphic window.

Color

MONO
A two color (black and white) file is saved.

GRAY
The specified file format is saved in gray scale.

COLOR
The file is saved at the specified color depth.

Krev

NORM
Saves file as shown on the screen.

REVERSE
Saves file with the background color reversed.

Orient

LANDSCAPE
Saves file in landscape mode.

PORTRAIT
Saves file in portrait mode.

Compress

YES
Compresses TIFF files and EPS files with TIFF preview (default).

NO
Saves files with no compression.

Quality

1 100
JPEG quality index, with 100 being the maximum quality level.

Notes

Allows you to activate specified GUI dialog boxes directly in either GUI or non-GUI mode.

The **/UI** command itself is valid in any processor, however certain dialog boxes are accessible only in a particular processor (e.g., **/UI**,RESULT,... is valid only in the General Postprocessor).

ANSYS JPEG software is based in part on the work of the Independent JPEG Group, Copyright 1998, Thomas G. Lane.

Menu Paths

This command cannot be accessed from a menu.

UIMP, *MAT*, *Lab1*, *Lab2*, *Lab3*, *VAL1*, *VAL2*, *VAL3*

Defines constant material properties (GUI).

PREP7: Materials

MP ME ST DY <> PR EM <> FL PP ED

MAT
Material number.

Lab1, *Lab2*, *Lab3*
Material property labels (see the **MP** command for valid labels).

VAL1, *VAL2*, *VAL3*
Values corresponding to three labels.

Notes

Defines constant material properties. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (**Jobname.LOG**) if material properties are specified using the Material Properties dialog box. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models
Main Menu>Preprocessor>Material Props>Material Models
Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

/UIS, *Label*, *VALUE*

Controls the GUI behavior.

SESSION: Run Controls

MP ME ST DY <> PR EM <> FL PP ED

Label
Behavior control key:

BORD
Controls the functionality of the mouse buttons for dynamic viewing mode only. When *Label* = BORD, the three values that follow control the functionality of the LEFT, MIDDLE and RIGHT buttons, respectively (see below).

MSGPOP
Controls which messages from the ANSYS error message subroutine are displayed in a message dialog box.

REPLOT
Controls whether or not an automatic replot occurs after functions affecting the model are executed.

ABORT
Controls whether or not ANSYS displays dialog boxes to show the status of an operation in progress and to cancel that operation.

DYNA
Controls whether the dynamic mode preview is a bounding box or the edge outline of the model. This label only applies to 2-D display devices (i.e., **/SHOW**,XII or **/SHOW**,WIN32). This "model edge outline" mode is only supported in PowerGraphics [**/GRAPHICS**,POWER] and is intended for element, line, results, area, or volume displays.

PICK
Controls how graphical entities are highlighted from within the ANSYS Select menu.

POWER
Controls whether or not PowerGraphics is active when the GUI is initiated. The ANSYS program default status is PowerGraphics "ON"; this command is used (placed in the **start.ans** file) when full graphics is desired on start up.

DPRO
Controls whether or not the ANSYS input window displays a dynamic prompt. The dynamic prompt shows the correct command syntax for the command, as you are entering it.

UNDO
Controls whether or not the session editor includes nonessential commands or comments in the file it creates. You can use this option to include comments and other materials in the session editor file.

LEGE
Controls whether or not the multi-legend is activated when you start the GUI. The multi-legend allows you specify the location of your legend items in each of the five graphics windows. You can place this option in your **start.ans** file and have the GUI start with the legend items in a pre-specified location.

PBAK
Controls whether or not the background shading is activated when you start the GUI. Background shading. You can place this option in your **start.ans** file and control whether or not background shading is activated.

ZPIC
Controls the sorting order for entities that are coincident (directly in front of or behind each other) to a picked spot on your model. When you pick a spot on your model that could indicate two or more entities, a message warns you of this condition, and a list of the coincident entities can be generated. The *VALUE* term (below) will determine the sort order.

HPOP
Controls the prioritization of your GUI windows when the contents are ported to a plot or print file (**/UI**,COPY,SAVE). OpenGL (3D) graphics devices require that the ANSYS Graphics Screen contents be set

in front of all overlying windows in order to port them to a printer or a file. This operation can sometimes conflict with **/NOERASE** settings. See the *VALUE* term (below) to determine the available control options.

VALUE

Values controlling behavior if *Label* = BORD:

(These values control the operation according to syntax : **/UIS**,BORD,*LEFT*,*MIDDLE*,*RIGHT*)

1
PAN, controls dynamic translations.

2
ZOOM, controls zoom, and dynamic rotation about the view vector.

3
ROTATE, controls dynamic rotation about the screen X and Y axes.

Note — You can designate any value for any button, or designate the same value for all three buttons. If no value is specified, default is LEFT = PAN, MIDDLE = ZOOM and RIGHT = ROTATE.

Values controlling behavior if *Label* = MSGPOP:

0
All messages displayed.

1
Only notes, warnings, and errors displayed.

2
Only warnings and errors displayed (default).

3
Only errors displayed.

Values controlling behavior if *Label* = REPLOT:

0
No automatic replot.

1
Automatic replot (default).

Values controlling behavior if *Label* = ABORT:

ON
Display status and cancellation dialog boxes (default).

OFF
Do not display status and cancellation dialog boxes.

1
Same as ON.

0
Same as OFF.

Values controlling behavior if *Label* = DYNA:

0
Use model edge outline when possible (default).

1
Use bounding box preview.

Values controlling behavior if *Label* = PICK:

0
Picked keypoints and nodes are enclosed by a square. Picked lines are overlaid by a thicker line. Picked areas, volumes, and elements (non-point/non-line) are redrawn with highlighting colors. However, if the pick is a box, circle, or polygon pick, the highlighting for all entitles consists only of a square placed around the entity's centroid.

1
Picked entities are not highlighted.

2
5.1 highlighting (that is, no XOR).

3
Picked entities are highlighted as in *VALUE* = 0, except that, for a box, circle, or polygon pick, the picked areas, volumes, and elements (non-point/non-line) are redrawn with highlighting colors. This technique is slower than the *VALUE* = 0 technique.

Values controlling behavior if *Label* = POWER:

0
Start GUI in Full Graphics mode.

1
Start GUI in PowerGraphics mode (default).

Values controlling behavior if *Label* = DPRO:

0 or OFF
Do not display the dynamic prompt.

1 or ON
Display the dynamic prompt (default).

Values controlling behavior if *Label* = UNDO:

0 or None
Do not suppress any commands (default).

1 or Comment
Write the nonessential commands to the session editor file as comments (with a ! at the beginning).

2 or Remove
Do not write nonessential commands or comments.

Values controlling behavior if *Label* = LEGE:

0 or OFF
Start GUI with the enhanced legend off (default).

1 or ON
Start GUI with the enhanced legend capability activated.

Values controlling behavior if *Label* = PBAK:

0 or OFF
Start the GUI with the no background shading (default).

1 or ON
Start the GUI with background shading activated.

Values controlling behavior if *Label* = HPOP:

0 or OFF
No rewrite operations are performed to compensate for items that obscure or overlay the graphics window (default).

1 or ON
The Graphics screen contents are replotted to ensure that they are situated in front of all other windows. If **/NOERASE** is detected, this operation is suppressed.

Notes

Controls certain features of the Graphical User Interface (GUI), including whether the ANSYS program displays dialog boxes to show you the status of an operation (such as meshing or solution) in progress and to enable you to cancel that operation. Issue **/UIS**,STAT for current status. Issue **/UIS**,DEFA to reset default values for all labels. Issue **/UIS**,*Label*,STAT and **/UIS**,*Label*,DEFA for status and to reset a specific *Label* item.

A **/UIS**,HPOP,1 command employs a fast redraw method which does not allow entering the legend logic for a **/PLOPTS**,INFO,1 or **/PLOPTS**,INFO,2 command. However, the legend *is* redrawn for **/PLOPTS**,INFO,3 because that command also allows a fast redraw.

This command is valid in any processor.

Menu Paths

Utility Menu>MenuCtrls>Message Controls
Utility Menu>PlotCtrls>Device Options

UNDELETE, *Option*, *Nstart*, *Nend*

Removes results sets from the group of sets selected for editing.

AUX3: Results Files

MP ME ST DY <> PR EM <> FL PP ED

Option
Specifies which sets are to be removed from the selected sets.

SET
Specifies one or more particular sets in the results file that are to be removed from the group of sets selected for deletion.

ALL
Removes all selected sets that are currently selected for deletion.

Nstart
The first set to be removed from the set selected for deletion.

Nend
The final set to be removed from the set selected for deletion. This field is used only if operating on more than one sequential set.

Notes

Use this command if you have previously marked a set for deletion (with the **DELETE** command) and now wish to keep that set instead of deleting it.

Menu Paths

This command cannot be accessed from a menu.

UNDO, *Kywrd*

Allows the user to modify or save commands issued since the last RESUME or SAVE command.

DATABASE: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Kywrd

NEW
Create an editable GUI window that allows the user to alter the commands issued since the most recent **SAVE** or **RESUME** operations (GUI only).

Notes

The **UNDO** command brings up the session editor, a text window that displays all of the program operations since the last **SAVE** or **RESUME** command. You can modify command parameters, delete whole sections of text and even save a portion of the command string to a separate file. The file is named **jobname000.cmds**, with each subsequent save operation incrementing the filename by one digit. For more information on the session editor, see Using the Session Editor in the *ANSYS Operations Guide*.

Menu Paths

Main Menu>Preprocessor>Session Editor

/UNITS, *Label*, *LENFACT*, *MASSFACT*, *TIMEFACT*, *TEMPFACT*, *TOFFSET*, *CHARGEFACT*, *FORCEFACT*, *HEATFACT*

Annotates the database with the system of units used.

DATABASE: Set Up

MP ME ST DY <> PR EM <> FL PP ED

Label

Label to denote the system of units used in this job:

USER
: User-defined system (default).

SI
: International system (SI or MKS; m, kg, s, K).

CGS
: CGS system (cm, g, s, °C).

MPA
: MPA system (mm, Mg, s, °C).

BFT
: British system using feet (ft, slug, s, °F).

BIN
: British system using inches (in, lbm, s, °F).

If *Label* = USER, the remaining fields on this command may be used to enter conversion factors that are appropriate for the user-defined system of units.

LENFACT
: Conversion factor to meter (m). Default = 1.

MASSFACT
: Conversion factor to kilogram (kg). Default = 1.

TIMEFACT
: Conversion factor to second (s). Default = 1.

TEMPFACT
: Conversion factor to Kelvin (K). Default = 1.

TOFFSET
: Temperature offset from absolute zero in degrees Kelvin. Default = 0.

CHARGEFACT
: Conversion factor to Coulomb. Default = 1.

FORCEFACT
: Conversion factor to Newton (N). Default = 1.

HEATFACT
: Conversion factor to Joule (J). Default = 1.

Command Default

User-defined units.

Notes

Allows the user to set a marker in the database indicating the system of units used. The setting may be reviewed with the **/STATUS** command at the Begin level. The units label and conversion factors on this command are for user convenience only and have no effect on the analysis or data. That is, **/UNITS** *will not* convert database items from one system to another (e.g., from British to SI, etc.). The units setting will be written to the file of IGES data [**IGESOUT** or **CDWRITE**], which can then be read by many programs that read IGES files. The user must still use consistent units for the results to be valid.

If you choose the MKS system of units, the EPZRO option for the **EMUNIT** command is set to 8.85 e-12 F/m. (EPZRO specifies alternate free-space permittivity.)

For micro-electromechanical systems (MEMS), where dimensions are on the order of microns, see the conversion factors in System of Units in the *ANSYS Coupled-Field Analysis Guide*.

If you use the ANSYS ADAMS Interface to export model information to the ADAMS program, the **/UNITS** command is required to ensure the correct transfer of data between ANSYS and ADAMS. You may choose a predefined unit system label (*Label* = SI, CGS, etc.) or you can select the user-defined system option (*Label* = USER) and input the appropriate conversion factors (*LENFACT*, *MASSFACT*, *TIMEFACT*, and *FORCEFACT*). The conversion factors will be written to the ADAMS input file **Jobname.MNF** in order to correctly generate the load. For more information, see Export to ADAMS in the *ANSYS Advanced Analysis Techniques Guide*.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

UPCOORD, *FACTOR*, *Key*

Modifies the coordinates of the active set of nodes, based on the current displacements.

SOLUTION: Load Step Options

MP ME ST <> <> PR <> <> <> PP ED

FACTOR

Scale factor for displacements being added to nodal coordinates. If *FACTOR* = 1.0, the full displacement value will be added to each node, 0.5, half the displacement value will be added, etc. If *FACTOR* = -1, the full displacement value will be subtracted from each node, etc.

Key

Key for zeroing displacements in the database:

OFF

Do not zero the displacements (default).

ON

Zero the displacements.

Notes

The **UPCOORD** command uses displacements stored in the ANSYS database, and not those contained within the results file, **Jobname.RST**. Nodal coordinates are updated each time the command is issued. After updating, both the nodal displacements and rotations are set to zero if *Key* = ON.

For structural solutions with an updated mesh, unless the coefficient matrix is otherwise reformed (e.g., a new analysis or **NLGEOM**,ON) it should first be reformed by issuing a **KUSE**,-1 command.

For a multiphysics simulation where a CFD or electromagnetic field is being coupled to a structure undergoing large displacements, all (or a portion) of the surrounding field mesh may take part in the structural solution to "move" with the displacing structure. You can use the **UPCOORD** command with a suitable *FACTOR* to update the coordinates of the nodes using the newly computed displacements. The mesh will now conform with the displaced structure for subsequent field solutions. However, the mesh should always be restored to its original location by using an **UPCOORD**,*FACTOR* command before performing any subsequent structural solutions. This is true for both repeated linear solutions, and for nonlinear restarts. (All saved displacements are relative to the original mesh location.)

This command is not intended to replace either the large displacement or birth and death logic.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Updt Node Coord
Main Menu>Solution>Load Step Opts>Other>Updt Node Coord

UPGEOM, *FACTOR*, *LSTEP*, *SBSTEP*, *Fname*, *Ext*, --

Adds displacements from a previous analysis and updates the geometry of the finite element model to the deformed configuration.

PREP7: Elements

MP ME ST DY <> PR EM <> FL PP ED

FACTOR
Multiplier for displacements being added to coordinates. The value 1.0 will add the full value of the displacements to the geometry of the finite element model. Defaults to 1.0.

LSTEP
Load step number of data to be imported. Defaults to the last load step.

SBSTEP
Substep number of data to be imported. Defaults to the last substep.

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The field must be input (no default).

Ext
Filename extension (8 character maximum).

The extension must be an RST extension.

- -

Unused field

Notes

This command updates the geometry of the finite element model according to the displacement results of the previous analysis and creates a revised geometry at the deformed configuration. This command works on all nodes (default) or on a selected set of nodes. If this command is issued repeatedly, it creates a revised geometry of the finite element model in a cumulative fashion, i.e., it adds displacement results on the previously generated deformed geometry. The solid model geometry is not updated by this command.

Menu Paths

Main Menu>Preprocessor>Modeling>Update Geom

/USER, *WN*

Conveniently resets /FOCUS and /DIST to USER.

GRAPHICS: Views

MP ME ST DY <> PR EM <> FL PP ED

WN

Window number (or ALL) to which command applies (defaults to 1).

Notes

Conveniently resets scale parameters to USER on the **/FOCUS** and **/DIST** commands. Scale parameters will be internally respecified to those used for the last display. Convenient when the last scale parameters were automatically calculated. User specified parameters hold until changed or removed [**/AUTO**]. Parameters may be reset on the individual commands after this command has been issued.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>View Settings>Automatic Fit Mode

USRCAL, *Rnam1*, *Rnam2*, *Rnam3*, *Rnam4*, *Rnam5*, *Rnam6*, *Rnam7*, *Rnam8*, *Rnam9*

Allows user-solution subroutines to be activated or deactivated.

SOLUTION: Load Step Options

MP ME ST <> <> <> <> <> <> PP <>

Rnam1, *Rnam2*, *Rnam3*, *Rnam4*, *Rnam5*, *Rnam6*, *Rnam7*, *Rnam8*, *Rnam9*

User-defined solution subroutine names to be activated. Up to nine may be defined on one command or multiple commands may be used. If *Rnam1* = ALL, activate all valid user subroutines. If *Rnam1* = NONE, deactivate all valid user subroutines. All characters are required:

USREFL
Allows user defined scalar field (body force) loads.

USERCV
Allows user defined convection (surface) loads.

USERPR
Allows user defined pressure (surface) loads.

USERFX
Allows user-defined heat flux (surface) loads.

USERCH
Allows user-defined charge density (surface) loads.

USERFD
Computes the complex load vector for the frequency domain logic (PIPE59).

USEROU
Allows user supplied element output.

USERMC
Allows user control of the hygrothermal growth (SHELL91).

USOLBEG
Allows user access before each solution.

ULDBEG
Allows user access before each load step.

USSBEG
Allows user access before each substep.

UITBEG
Allows user access before each equilibrium iteration.

UITFIN
Allows user access after each equilibrium iteration.

USSFIN
Allows user access after each substep.

ULDFIN
Allows user access after each load step.

USOLFIN
Allows user access after each solution.

UANBEG
Allows user access at start of ANSYS run.

UANFIN
Allows user access at end of ANSYS run.

UELMATX
Allows user access to element matrices and load vectors.

Command Default

No user-solution subroutines are active (even if linked into the program).

Notes

Allows certain user-solution subroutines to be activated or deactivated (system-dependent). This command only affects the subroutines named. Other user subroutines (such as user elements, user creep, etc.) have their own activation controls described with the feature.

The routines are commented and should be listed from the distribution media (system-dependent) for more details. See also the *ANSYS Advanced Analysis Techniques Guide* for a general description of user-programmable features.

Users must have system permission, system access, and knowledge to write, compile, and link the appropriate subroutines into the ANSYS program at the site where it is to be run. All routines should be written in FORTRAN 77. Issue **USRCAL**,STAT to list the status of these user subroutines. Since a user-programmed subroutine is a nonstandard use of the program, the verification of any ANSYS run incorporating these commands is entirely up to the user. In any contact with ANSYS customer support regarding the performance of a custom version of the ANSYS program, you should explicitly state that a user programmable feature has been used.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>User Routines
Main Menu>Solution>Load Step Opts>Other>User Routines

V Commands

V, *P1*, *P2*, *P3*, *P4*, *P5*, *P6*, *P7*, *P8*
Defines a volume through keypoints.

PREP7: Volumes

MP ME ST DY <> PR EM EH FL PP ED

P1
Keypoint defining starting corner of volume. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P2
Keypoint defining second corner of volume.

P3
Keypoint defining third corner of volume.

P4
Keypoint defining fourth corner of volume.

P5
Keypoint defining fifth corner of volume.

P6
Keypoint defining sixth corner of volume.

P7
Keypoint defining seventh corner of volume.

P8
Keypoint defining eighth corner of volume.

Notes

Defines a volume (and its corresponding lines and areas) through eight (or fewer) existing keypoints. Keypoints must be input in a continuous order. The order of the keypoints should be around the bottom and then the top. Missing lines are generated "straight" in the active coordinate system and assigned the lowest available numbers [**NUMSTR**]. Missing areas are generated and assigned the lowest available numbers.

Note — Solid modeling in a toroidal coordinate system is not recommended.

Certain faces may be condensed to a line or point by repeating keypoints. For example, use **V**,*P1*,*P2*,*P3*,*P3*,*P5*,*P6*,*P7*,*P7* for a triangular prism or **V**,*P1*,*P2*,*P3*,*P3*,*P5*,*P5*,*P5*,*P5* for a tetrahedron. See Figure 45.1: "SOLID45 Geometry" in the *ANSYS Elements Reference* for analogous element sketches of the valid condensed face patterns.)

Using keypoints to produce partial sections in **CSYS** = 2 can generate anomalies; check the resulting volumes carefully.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Arbitrary>Through KPs

V2DOPT, *GEOM*, *NDIV*, *HIDOPT*, *NZONE*

Specifies 2-D/axisymmetric view factor calculation options.

SOLUTION: Radiosity

MP ME <> <> <> PR <> <> <> PP ED

GEOM
Choice of geometry:

0
Planar (default).

1
Axisymmetric

NDIV
Number of divisions for axisymmetric geometry. Defaults to 20.

HIDOPT
Viewing option:

0
Hidden (default).

1
Non-hidden

NZONE
Number of zones for view factor calculation. Defaults to 200.

Command Default

V2DOPT allows you to select options for 2-D view factor calculation. The geometry type can be set to either 2-D plane or axisymmetric (defaults to plane). You can also define the number of divisions (defaults to 20) for an axisymmetric geometry. This command also allows you to select either hidden or non-hidden viewing option (defaults to hidden) and the number of zones for view factor calculation (defaults to 200).

Menu Paths

Main Menu>Preprocessor>Radiation Opts>View Factor
Main Menu>Radiation Opt>Radiosity Meth>View Factor
Main Menu>Solution>Radiation Opts>View Factor

VA, *A1*, *A2*, *A3*, *A4*, *A5*, *A6*, *A7*, *A8*, *A9*, *A10*

Generates a volume bounded by existing areas.

PREP7: Volumes

MP ME ST DY <> PR EM EH FL PP ED

A1, *A2*, *A3*, *A4*, *A5*, *A6*, *A7*, *A8*, *A9*, *A10*
List of areas defining volume. The minimum number of areas is 4. If *A1* = ALL, use all selected [**ASEL**] areas and ignore *A2* to *A10*. If *A1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *A1*.

Notes

This command conveniently allows generating volumes from regions having more than eight keypoints (which is not allowed with the **V** command). Areas may be input in any order. The exterior surface of a **VA** volume must be continuous, but holes may pass completely through it.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volume by Areas
Main Menu>Preprocessor>Modeling>Create>Volumes>Arbitrary>By Areas
Main Menu>Preprocessor>Modeling>Geom Repair>Create Volume

VADD, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Adds separate volumes to create a single volume.

PREP7: Booleans

MP ME ST DY <> PR EM EH FL PP ED

NV1, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*
Numbers of volumes to be added. If *NV1* = ALL, add all selected volumes and ignore *NV2* to *NV9*. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Adds separate volumes to create a single volume. The original volumes (and their corresponding areas, lines and keypoints) will be deleted by default [**BOPTN**]. See the **BOPTN** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. Concatenated entities are not valid with this command.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Add>Volumes
Main Menu>Preprocessor>Modeling>Operate>Booleans>Add>Volumes

VALVE, *NLOC*, *LENG*, *MASS*, *SIF*, *FLEX*, *ARINS*, *ELEM*

Defines a valve in a piping run.

PREP7: Piping

MP ME ST <> <> PR <> <> <> PP ED

NLOC
Node where valve is to be placed (as described below). Defaults to current run starting point.

LENG
Length of valve (defaults to larger pipe OD).

MASS
Dry mass (weight/gravity) of valve without insulation (defaults to equivalent straight pipe mass). Note, acceleration [**ACEL**] must be nonzero for weight to be calculated.

SIF
Stress intensification factor (defaults to 1.0).

FLEX
Bending flexibility factor (defaults to 0.5).

ARINS
Insulation surface area (defaults to equivalent straight pipe insulation area). Units (length^2) must be consistent with the smallest unit of the system used (not mixed) regardless of the **PUNIT** option.

ELEM
Element number to be assigned to valve (defaults to the previous maximum element number (MAXEL) + 1).

Notes

Defines a valve (straight pipe element (PIPE16) with adjusted specifications and loadings) at a given location in a piping run. See the PREP7 **RUN** command. The location may be 1) between two adjacent colinear straight pipes, 2) between an adjacent straight pipe and a different piping component, or 3) at the end of a straight pipe.

For Case 1, two new nodes are generated at the ends of the valve. The two straight pipes are automatically "shortened" to meet the ends of the valve. The valve specifications and loadings are taken from the corresponding two straight pipes.

For Case 2, one new node is generated at one end of the valve. The straight pipe is automatically "shortened" to meet this end of the valve. The other end of the valve meets the other piping component. The valve specifications and loadings are taken from the straight pipe.

For Case 3, one new node is generated at the free end of the valve. The other end of the valve meets the straight pipe. The valve specifications and loadings are taken from the straight pipe.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Piping Models>Define Pipes>Valve

---

VARDEL, *NVAR*
Deletes a variable (GUI).

POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

NVAR
The reference number of the variable to be deleted. *NVAR* is as defined by **NSOL**, **ESOL**, etc.

Notes

Deletes a POST26 solution results variable. This is a command generated by the Graphical User Interface (GUI). It will appear in the log file (**Jobname.LOG**) if a POST26 variable is deleted from the "Defined Time-History Variables" dialog box. This command is *not* intended to be typed in directly in an ANSYS session (although it can be included in an input file for batch input or for use with the **/INPUT** command).

Menu Paths

Main Menu>TimeHist Postpro>Define Variables
Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

VARNAM, *IR*, *Name*

Names (or renames) a variable.

POST26: Set Up

MP ME ST DY <> PR EM <> FL PP ED

IR
Reference number of the variable (2 to NV [**NUMVAR**]).

Name
Thirty-two character name for identifying variable on printouts and displays. Embedded blanks are compressed for output.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph
Main Menu>TimeHist Postpro>Settings>List

VATT, *MAT*, *REAL*, *TYPE*, *ESYS*

Associates element attributes with the selected, unmeshed volumes.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

MAT, *REAL*, *TYPE*, *ESYS*
Material number, real constant set number, type number, and coordinate system number to be associated with selected, unmeshed volumes.

Notes

These element attributes will be used when the volumes are meshed. If a volume does not have attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT**, **REAL**, **TYPE**, and **ESYS** command settings. Reissue the **VATT** command (before volumes are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association.

If any of the arguments *MAT*, *REAL*, *TYPE*, or *ESYS* are defined as -1, then that value will be left unchanged in the selected set.

In some cases, ANSYS can proceed with a volume meshing operation even when no logical element type has been assigned via **VATT**,,,*TYPE* or **TYPE**. For more information, see the discussion on setting element attributes in Meshing Your Solid Model of the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>All Volumes

Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked Volumes

---

VCLEAR, *NV1*, *NV2*, *NINC*

Deletes nodes and volume elements associated with selected volumes.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

NV1, *NV2*, *NINC*

Delete mesh for volumes *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and mesh for all selected volumes [**VSEL**] is deleted. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

Notes

Deletes *all* nodes and volume elements associated with selected volumes (regardless of whether the nodes or elements are selected). Nodes shared by adjacent meshed volumes and nodes associated with non-volume elements will not be deleted. Attributes assigned as a result of **VATT** are maintained. In the program's response to the command, if a volume, area, line, or keypoint is tallied as "cleared," it means either its node or element reference was deleted.

Menu Paths

Main Menu>Preprocessor>Meshing>Clear>Volumes

/VCONE, *WN*, *PHI*

Defines the view cone angle for perspective displays.

GRAPHICS: Views

MP ME ST DY <> PR EM <> FL PP ED

WN

Window number (or ALL) to which command applies (defaults to 1).

PHI

View cone angle (0.0 to 85.°) to define perspective. Use *PHI* = 45.0° for typical perspective. Increase angle for more perspective, decrease angle for less. If the distance [**/DIST**] is not specified, it will be automatically calculated to give full window magnification. If the distance is also specified, *PHI* controls both the perspective and the magnification. The larger the angle, the more the perspective and the less the magnification. Defaults to 0.0 (no perspective).

Command Default

0.0 degrees (no perspective (parallel projection)).

Notes

Perspective shows the true depth of the object in the display. A variable magnification results since the back plane of the object is further from the observer than the front plane. The largest magnification occurs at the front plane. With perspective, the magnification factor (MAGF) is not only a function of the distance from the object, but also the window shape and the perspective (or view cone) angle Φ as follows:

$$MAGF = \frac{\ell/2}{(d)\,TAN\phi}$$

where ℓ, for square windows, is the largest in-plane vertical or horizontal dimension, d is the distance from the observer to the plane of ℓ (usually the front plane of the object), and Φ is the view cone angle (defined with the **/VCONE** command). The bigger the cone angle, the more the perspective. The magnification factor proportionally decreases with increasing Φ. The distance can be defined with the **/DIST** or the **/FOCUS** command. Note, the distance input on the **/DIST** command is equal to d only if the focus point is located on the plane of ℓ. It is recommended that if a general perspective is desired (i.e., not any specific cone angle), use Φ = 45.0 (since TAN(45.0) = 1.0) and let the d value be automatically calculated for full window magnification.

Note that any number of **/DIST**, **/FOCUS**, and **/VCONE** combinations can be used to produce the same magnification. Distances less than the object depth will produce views from within the object.

A magnification factor of 1.0 just fills the window. If the automatic scaling option is used [**/AUTO**], the magnification factor is fixed at 0.91 (to allow a 10% margin around the object) and d is automatically calculated for the given **/VCONE** and **/FOCUS** values. Any value of Φ between 0.0 and 85.0 (usually 45.0) may be used to activate the perspective. Views from inside the object are not possible when d is automatically calculated (use manual scaling [**/USER**] along with **/DIST** specification).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>View Settings>Perspective View

VCROSS, *LabXR*, *LabYR*, *LabZR*, *LabX1*, *LabY1*, *LabZ1*, *LabX2*, *LabY2*, *LabZ2*

Forms element table items from the cross product of two vectors.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

LabXR, *LabYR*, *LabZR*
Label assigned to X, Y, and Z-component of resultant vector.

LabX1, *LabY1*, *LabZ1*
X, Y, and Z-component of first vector label.

LabX2, *LabY2*, *LabZ2*
X, Y, and Z-component of second vector label.

Notes

Forms labeled result items for the selected element from the cross product of two vectors:

{*LabXR*, *LabYR*, *LabZR*} = {*LabX1*, *LabY1*, *LabZ1*} X {*LabX2*, *LabY2*, *LabZ2*}

Data must be in a consistent coordinate system. Labels are those associated with the **ETABLE** command.

Menu Paths

Main Menu>General Postproc>Element Table>Cross Product

VCVFILL, *A1*, *A2*, *A3*, *A4*, *A5*, *A6*, *A7*, *A9*, *A9*, *A10*

Fills cavities and bosses in volumes (for models imported from CAD files).

PREP7: CAD Repair

MP ME ST DY <> PR EM EH FL PP ED

A1, *A2*, *A3*, *A4*, *A5*, *A6*, *A7*, *A9*, *A9*, *A10*
List of areas that define the volume. If *A1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). If *A1* = ALL, all selected areas will be used and all remaining arguments are ignored.

Notes

Use this command to fill in holes and cavities or eliminate protrusions (bosses) in models imported from CAD files. In essence, this command removes features projecting above or below a plane, such as bosses and cavities. This command is available only for models imported through the Default IGES option.

You must select all areas related to the feature before it can be eliminated, and the order in which those areas are selected can be important. If the boss or cavity is an isolated entity, the area selection order does not matter. However, if the boss or cavity is attached to an area, that area must be the last area selected in the list. If the attached area is not the last area in the selection list the command may distort the geometry of the model.

Menu Paths

Main Menu>Preprocessor>Modeling>Simplify>Toolkit>Fill Cavity
Main Menu>Preprocessor>Modeling>Simplify>Toolkit>Remove Boss

VDDAM, *VF*, *VA*, *VB*, *VC*

Specifies the velocity spectrum computation constants for the analysis of shock resistance of shipboard structures.

SOLUTION: Spectrum Options

MP ME ST <> <> <> <> <> <> PP ED

VF
Direction-dependent velocity coefficient for elastic or elastic-plastic analysis option (Default = 0).

VA, *VB*, *VC*
Coefficients for the DDAM velocity spectrum equations. See the *ANSYS, Inc. Theory Reference*. Default for these coefficients is zero.

Notes

This command specifies velocity coefficients to analyze shock resistance of shipboard equipment. These coefficients are used to compute mode coefficients according to the equations given in the *ANSYS, Inc. Theory Reference*. The form of these equations is based on the Naval NRL Dynamic Design Analysis Method. This command, along with the **ADDAM** and **SED** commands, is used with the spectrum (**ANTYPE**,SPECTR) analysis as a special purpose alternative to the **SV**, **FREQ**, and **SVTYP** commands. The mass and length units of the model must be in pounds and inches, respectively.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>DDAM Options
Main Menu>Solution>Load Step Opts>Spectrum>DDAM Options

VDELE, *NV1*, *NV2*, *NINC*, *KSWP*

Deletes unmeshed volumes.

PREP7: Volumes

MP ME ST DY <> PR EM <> FL PP ED

NV1, *NV2*, *NINC*

Delete volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and all selected volumes [**VSEL**] are deleted. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

KSWP

Specifies whether keypoints, lines, and areas are also deleted:

0

Delete volumes only (default).

1

Delete volumes, as well as keypoints, lines, and areas attached to the specified volumes but not shared by other volumes.

Menu Paths

Main Menu>Preprocessor>Modeling>Delete>Volume and Below
Main Menu>Preprocessor>Modeling>Delete>Volumes Only
Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Delete>Volume and Below
Main Menu>Preprocessor>Trefftz Domain>TZ Geometry>Delete>Volumes Only

---

VDGL, *NV1*, *NV2*, *NINC*

Lists keypoints of a volume that lie on a parametric degeneracy.

PREP7: Volumes

MP ME ST DY <> PR EM <> FL PP ED

NV1, *NV2*, *NINC*

List keypoints that lie on a parametric degeneracy on volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL (default), *NV2* and *NINC* will be ignored and keypoints on all selected volumes [**VSEL**] will be listed. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). You may also substitute a component name for *NV1* (ignore *NV2* and *NINC*).

Notes

See the *ANSYS Modeling and Meshing Guide* for details about parametric degeneracies.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>List Degen Volus
Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>List Degen Volus

---

VDOT, *LabR*, *LabX1*, *LabY1*, *LabZ1*, *LabX2*, *LabY2*, *LabZ2*

Forms an element table item from the dot product of two vectors.

POST1: Element Table

MP ME ST DY <> PR EM <> FL PP ED

LabR

Label assigned to dot product result.

LabX1, *LabY1*, *LabZ1*

X, Y, and Z-component of first vector label.

LabX2, *LabY2*, *LabZ2*

X, Y, and Z-component of second vector label.

Notes

Forms labeled result items for the selected element from the dot product of two vectors:

$$LabR = \{LabX1, LabY1, LabZ1\} \bullet \{LabX2, LabY2, LabZ2\}$$

Data must be in a consistent coordinate system. Labels are those associated with the **ETABLE** command.

Menu Paths

Main Menu>General Postproc>Element Table>Dot Product

VDRAG, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NLP1*, *NLP2*, *NLP3*, *NLP4*, *NLP5*, *NLP6*
Generates volumes by dragging an area pattern along a path.

PREP7: Volumes

MP ME ST DY <> PR EM EH FL PP ED

NA1, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*
List of areas in the pattern to be dragged (6 maximum if using keyboard entry). If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NA1* = ALL, all selected areas will be swept along the path. A component name may also be substituted for *NA1*.

NLP1, *NLP2*, *NLP3*, *NLP4*, *NLP5*, *NLP6*
List of lines defining the path along which the pattern is to be dragged (6 maximum if using keyboard entry). Must be a continuous set of lines. To be continuous, adjacent lines must share the connecting keypoint (the end keypoint of one line must also be first keypoint of the next line).

Notes

Generates volumes (and their corresponding keypoints, lines, and areas) by sweeping a given area pattern along a characteristic drag path. If the drag path consists of multiple lines, the drag direction is determined by the sequence in which the path lines are input (*NLP1*, *NLP2*, etc.). If the drag path is a single line (*NLP1*), the drag direction is from the keypoint on the drag line that is closest to the first keypoint of the given area pattern to the other end of the drag line.

The magnitude of the vector between the keypoints of the given pattern and the first path keypoint remains constant for all generated keypoint patterns and the path keypoints. The direction of the vector relative to the path slope also remains constant so that patterns may be swept around curves. Lines are generated with the same shapes as the given pattern and the path lines.

Keypoint, line, area, and volume numbers are automatically assigned (beginning with the lowest available values [**NUMSTR**]). Adjacent lines use a common keypoint, adjacent areas use a common line, and adjacent volumes use a common area. For best results, the entities to be dragged should be orthogonal to the start of the drag path. Drag operations that produce an error message may create some of the desired entities prior to terminating.

If element attributes have been associated with the input area via the **AATT** command, the opposite area generated by the **VDRAG** operation will also have those attributes (i.e., the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the input areas are meshed or belong to a meshed volume, the area(s) can be extruded to a 3-D mesh. Note that the *NDIV* argument of the **ESIZE** command should be set before extruding the meshed areas. Alternatively, mesh divisions can be specified directly on the drag line(s) (**LESIZE**). See the *ANSYS Modeling and Meshing Guide* for more information.

You can use the **VDRAG** command to generate 3-D interface element meshes for elements INTER194 and INTER195. When generating interface element meshes using **VDRAG**, you must specify the line divisions to generate one interface element directly on the drag line using the **LESIZE** command. The source area to be extruded becomes the bottom surface of the interface element. Interface elements must be extruded in what will become the element's local x direction, that is, bottom to top.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>Along Lines

VEXT, *NA1*, *NA2*, *NINC*, *DX*, *DY*, *DZ*, *RX*, *RY*, *RZ*

Generates additional volumes by extruding areas.

PREP7: Volumes

MP ME ST DY <> PR EM EH FL PP ED

NA1, *NA2*, *NINC*
: Set of areas (*NA1* to *NA2* in steps of *NINC*) that defines the pattern to be extruded. *NA2* defaults to *NA1*, *NINC* defaults to 1. If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is defined by all selected areas. If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

DX, *DY*, *DZ*
: Increments to be applied to the X, Y, and Z keypoint coordinates in the active coordinate system (*DR*, *D*θ, *DZ* for cylindrical; *DR*, *D*θ, *D*Φ for spherical).

RX, *RY*, *RZ*
: Scale factors to be applied to the X, Y, and Z keypoint coordinates in the active coordinate system (*RR*, *R*θ, *RZ* for cylindrical; *RR*, *R*θ, *R*Φ for spherical). Note that the *R*θ and *R*Φ scale factors are interpreted as angular *offsets*. For example, if CSYS = 1, *RX*, *RY*, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

Notes

Generates additional volumes (and their corresponding keypoints, lines, and areas) by extruding and scaling a pattern of areas in the active coordinate system.

If element attributes have been associated with the input area via the **AATT** command, the opposite area generated by the **VEXT** operation will also have those attributes (i.e., the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the areas are meshed or belong to meshed volumes, a 3-D mesh can be extruded with this command. Note that the *NDIV* argument on the **ESIZE** command should be set before extruding the meshed areas.

Scaling of the input areas, if specified, is performed first, followed by the extrusion.

In a non-Cartesian coordinate system, the **VEXT** command locates the end face of the volume based on the active coordinate system. However, the extrusion is made along a straight line between the end faces. Note that solid modeling in a toroidal coordinate system is not recommended.

Caution: Use of the **VEXT** command can produce unexpected results when operating in a non-Cartesian coordinate system. For a detailed description of the possible problems that may occur, see Solid Modeling in the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>By XYZ Offset

VFCALC, *Fname*, *Ext*, --

Computes and stores Hemicube view factors.

SOLUTION: Radiosity

MP ME <> <> <> PR <> <> <> PP ED

Fname
File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to **Jobname**.

Ext
Filename extension (8 character maximum).

The extension defaults to VF.

--
Unused field

Notes

When you leave ANSYS, the Hemicube view factors are removed from memory.

Menu Paths

Main Menu>Radiation Opt>Radiosity Meth>Compute

VFOPT, *Opt*, *Filename*, *Ext*, *Dir*, *Format*

Specifies options for view factor file.

SOLUTION: Radiosity

MP ME <> <> <> PR <> <> <> PP ED

Opt
View factor option:

NEW
Compute view factors and write them to a file.

OFF
Do not recompute view factors it they already exist in the database, otherwise compute them (default).

READ
Read view factors from a file. For subsequent **SOLVE** commands, switch to the default option (OFF).

Fname
File name for view factor matrix. Defaults to **Jobname**.

Ext
Filename extension for view factor matrix (default = vf).

Dir
Directory path for view factor matrix. If you do not specify a directory path, it will default to your working directory.

Format
File format for view factor file. Defaults to a Binary format.

BINA
Binary file (default).

ASCI
ASCII file.

Notes

This command allows you to deactivate the view factor computation (*Opt* = OFF) if the view factors already exist in the database. OFF is the default upon encountering the second and subsequent **SOLVE** commands in the solution processor.

Menu Paths

Main Menu>Preprocessor>Radiation Opts>View Factor
Main Menu>Radiation Opt>Radiosity Meth>View Factor
Main Menu>Solution>Radiation Opts>View Factor

VFQUERY, *SRCELEM*, *TARELEM*

Queries and prints element Hemicube view factors and average view factor.

AUX12: Radiation Substructures

MP ME <> <> <> PR <> <> <> PP ED

SRCELEM
Elements representing the source radiating surfaces used to query the view factor at the target element(s). If *SRCELEM* = P, graphical picking is enabled (valid only in the GUI). If *SRCELEM* = ALL, all selected elements will have their view factors queried. A component name may also be substituted for *SRCELEM*. Selected elements must be flagged for surface to surface radiation in order to query view factors (**SF**, **SFA**, or **SFE** with Lab = RDSF). The view factors must have been previously computed.

TARELEM
Element for view factor query. If *TARELEM* = P, graphical picking is enabled (valid only in the GUI). If *TARELEM* = ALL, all selected elements will have their view factors queried. A component name may also be substituted for *TARELEM*. Selected elements must be flagged for surface to surface radiation in order to query view factors (**SF**, **SFA**, or **SFE** with Lab = RDSF). The view factors must have been previously computed.

Notes

View factors for each target element will be printed.

An average view factor for all target elements will be computed. (Use ***GET** to retrieve the average value).

When resuming a database, issue the command **VFOPT**,READ before issuing the **VFQUERY** command.

Menu Paths

Main Menu>Radiation Opt>Radiosity Meth>Query

VGEN, *ITIME*, *NV1*, *NV2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

Generates additional volumes from a pattern of volumes.

PREP7: Volumes

MP ME ST DY <> PR EM EH FL PP ED

ITIME

Do this generation operation a total of *ITIME*s, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be > 1 for generation to occur.

NV1, *NV2*, *NINC*

Generate volumes from pattern beginning with *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is all selected volumes [**VSEL**]. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

DX, *DY*, *DZ*

Keypoint location increments in the active coordinate system (--, Dθ, DZ for cylindrical, --, Dθ, -- for spherical).

KINC

Keypoint increment between generated sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM

Specifies if elements and nodes are also to be generated:

0

Generate nodes and elements associated with the original volumes, if they exist.

1

Do not generate nodes and elements.

IMOVE

Specifies whether to redefine the existing volumes:

0

Generate additional volumes as requested with the *ITIME* argument.

1

Move original volumes to new position retaining the same keypoint line, and area numbers (*ITIME*, *KINC*, and *NOELEM* are ignored). Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates additional volumes (and their corresponding keypoints, lines, areas and mesh) from a given volume pattern. The MAT, TYPE, REAL, and ESYS attributes are based upon the volumes in the pattern and not upon the current settings of the pointers. End slopes of the generated lines remain the same (in the active coordinate system) as those of the given pattern. For example, radial slopes remain radial, etc. Generations which produce volumes of a size or shape different from the pattern (i.e., radial generations in cylindrical systems, radial and phi

generations in spherical systems, and theta generations in elliptical systems) are not allowed. Note that solid modeling in a toroidal coordinate system is not recommended. Volume, area, and line numbers are automatically assigned (beginning with the lowest available values [**NUMSTR**]).

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Volumes
Main Menu>Preprocessor>Modeling>Move / Modify>Volumes

VGET, *Par*, *IR*, *TSTRT*, *KCPLX*

Moves a variable into an array parameter vector.

POST26: Special Purpose

MP ME ST DY <> PR EM <> FL PP ED

Par
Array parameter vector in the operation.

IR
Reference number of the variable (1 to NV [**NUMVAR**]).

TSTRT
Time (or frequency) corresponding to start of *IR* data. If between values, the nearer value is used.

KCPLX
Complex number key:

0
Use the real part of the *IR* data.

1
Use the imaginary part of the *IR* data.

Notes

Moves a variable into an array parameter vector. The starting array element number must be defined. For example, **VGET**,A(1),2 moves variable 2 (starting at time 0.0) to array parameter A. Looping continues from array element A(1) with the index number incremented by one until the variable is filled. The number of loops may be controlled with the ***VLEN** command (except that loop skipping (*NINC*) is not allowed). For multi-dimensioned array parameters, only the first (row) subscript is incremented.

Menu Paths

Main Menu>TimeHist Postpro>Table Operations>Variable to Par

VGLUE, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Generates new volumes by "gluing" volumes.

PREP7: Booleans

MP ME ST DY <> PR EM EH FL PP ED

NV1, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Numbers of the volumes to be glued. If *NV1* = ALL, all selected volumes will be glued (*NV2* to *NV9* will be ignored). If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Use of the **VGLUE** command generates new volumes by "gluing" input volumes. The glue operation redefines the input volumes so that they share areas along their common boundaries. The new volumes encompass the same geometry as the original volumes. This operation is only valid if the intersections of the input volumes are areas along the boundaries of those volumes. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

The **VGLUE** command results in the merging of areas, lines, and keypoints at the common volume boundaries. The areas, lines, and keypoints of the lower numbered volume will be kept. This means one must be aware of volume numbering when multiple **VGLUE** commands are applied to avoid any "ungluing" of geometry.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Glue>Volumes

---

/VIEW, *WN*, *XV*, *YV*, *ZV*

Defines the viewing direction for the display.

GRAPHICS: Views

MP ME ST DY <> PR EM <> FL PP ED

WN

Window number (or ALL) to which command applies (defaults to 1).

XV, *YV*, *ZV*

The object is viewed along the line from point *XV*,*YV*,*ZV* (in the global coordinate system) to the global coordinate system origin. For section displays, the cutting plane is assumed to be perpendicular to this line. If *XV* = WP, modify view to be normal to the currently defined working plane. Defaults to (0,0,1).

Command Default

0,0,1 view. The default reference orientation is X-axis horizontal to the right, Y-axis vertical upward, and Z-axis out from the screen (normal). See **/VUP** command to change reference orientation.

Notes

The view line is always normal to the screen. The view is selected by defining a point (in the global Cartesian coordinate system) representing a point along the viewing line. This point, and the global Cartesian coordinate system origin, define the line along which the object is viewed while looking toward the origin. Any point along the view line may be used, i.e., (1,1,1) and (2,2,2) give the same view. The display orientation may be changed as desired [**/ANGLE**]. The display coordinate system type may be changed (from Cartesian to cylindrical, spherical, toroidal, etc.) with the **DSYS** command.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane
Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane
Utility Menu>PlotCtrls>Pan, Zoom, Rotate
Utility Menu>PlotCtrls>View Settings>Viewing Direction

VIMP, *VOL*, *CHGBND*, *IMPLEVEL*

Improves the quality of the tetrahedral elements in the selected volume(s).

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

VOL
Number of the volume containing the tetrahedral elements to be improved. If *VOL* = ALL (default), improve the tetrahedral elements in all selected volumes. If *VOL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *VOL*.

CHGBND
Specifies whether to allow boundary modification. Boundary modification includes such things as changes in the connectivity of the element faces on the boundary and the addition of boundary nodes. (Also see Notes below for important usage information for *CHGBND*.)

0
Do not allow boundary modification.

1
Allow boundary modification (default).

IMPLEVEL
Identifies the level of improvement to be performed on the elements. (Improvement occurs primarily through the use of face swapping and node smoothing techniques.)

0
Perform the least amount of swapping/smoothing.

1
Perform an intermediate amount of swapping/smoothing.

2
Perform the greatest amount of swapping/smoothing.

3
Perform the greatest amount of swapping/smoothing, plus additional improvement techniques (default).

Notes

VIMP is useful for further improving a volume mesh created in ANSYS [**VMESH**], especially quadratic tetrahedral element meshes.

The **VIMP** command enables you to improve a given tetrahedral mesh by reducing the number of poorly-shaped tetrahedral elements (in particular, the number of sliver tetrahedral elements)--as well as the overall number of elements--in the mesh. It also improves the overall quality of the mesh.

Regardless of the value of the *CHGBND* argument, boundary mid-nodes can be moved as long as you are not using p-method analysis. When *CHGBND* = 0 and you are using p-method analysis, boundary mid-nodes cannot be moved. (ANSYS issues an error message if it would be necessary to move boundary mid-nodes in order to generate valid quadratic elements.)

When loads or constraints have been placed on boundary nodes or mid-nodes, and boundary mid-nodes are later moved, ANSYS issues a warning message to let you know that it will not update the loads or constraints.

Even when *CHGBND* = 1, no boundary modification is performed on areas and lines that are not modifiable (for example, areas that are adjacent to other volumes or that contain shell elements, or lines that are not incident on modifiable areas, contain beam elements, or have line divisions specified for them [**LESIZE**]).

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Improve Tets>Volumes

---

VINP, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Finds the pairwise intersection of volumes.

PREP7: Booleans

MP ME ST DY <> PR EM EH FL PP ED

NV1, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*
Numbers of volumes to be intersected pairwise. If *NV1* = ALL, *NV2* to *NV9* are ignored and the pairwise intersection of all selected volumes is found. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Finds the pairwise intersection of volumes. The pairwise intersection is defined as all regions shared by any two or more volumes listed on this command. New volumes will be generated where the original volumes intersect pairwise. If the regions of pairwise intersection are only areas, new areas will be generated. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Pairwise>Volumes

---

VINV, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*
Finds the intersection of volumes.

PREP7: Booleans

MP ME ST DY <> PR EM EH FL PP ED

NV1, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*
Numbers of volumes to be intersected. If *NV1* = ALL, *NV2* to *NV9* are ignored, and the intersection of all selected volumes is found. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Finds the common (not pairwise) intersection of volumes. The common intersection is defined as the regions shared (in common) by **all** volumes listed on this command. New volumes will be generated where the original volumes intersect. If the regions of intersection are only areas, new areas will be generated instead. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Common>Volumes

---

VLIST, *NV1*, *NV2*, *NINC*
Lists the defined volumes.

PREP7: Volumes

MP ME ST DY <> PR EM <> FL PP ED

NV1, *NV2*, *NINC*
List volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL (default), *NV2* and *NINC* are ignored and all selected volumes [**VSEL**] are listed. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

Notes

An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **VATT** command (and will not be reset to zero if the mesh is cleared); one listed as a negative value indicates that the attribute was assigned using the attribute pointer [TYPE, MAT, REAL, or ESYS] that was active during meshing (and will be reset to zero if the mesh is cleared). A "-1" in the "nodes" column indicates that the volume has been meshed but there are no interior nodes. The volume size is listed only if a **VSUM** command has been performed on the volume.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Volumes

VLSCALE, *NV1*, *NV2*, *NINC*, *RX*, *RY*, *RZ*, *KINC*, *NOELEM*, *IMOVE*

Generates a scaled set of volumes from a pattern of volumes.

PREP7: Volumes

MP ME ST DY <> PR EM EH FL PP ED

NV1, *NV2*, *NINC*

Set of volumes (*NV1* to *NV2* in steps of *NINC*) that defines the pattern to be scaled. *NV2* defaults to *NV1*, *NINC* defaults to 1. If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is defined by all selected volumes. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

RX, *RY*, *RZ*

Scale factors to be applied to the X, Y, and Z keypoint coordinates in active coordinate system (*RR*, *R*θ, *RZ* for cylindrical; *RR*, *R*θ, *R*Φ for spherical). Note that the *R*θ and *R*Φ scale factors are interpreted as angular *offsets*. For example, if CSYS = 1, *RX*, *RY*, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

KINC

Increment to be applied to keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned [**NUMSTR**].

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Nodes and elements associated with the original volumes will be generated (scaled) if they exist.

1

Nodes and elements will *not* be generated.

IMOVE

Specifies whether volumes will be moved or newly defined:

0

Additional volumes will be generated.

1

Original volumes will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old volumes are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a scaled set of volumes (and their corresponding keypoints, lines, areas, and mesh) from a pattern of volumes. The MAT, TYPE, REAL, and ESYS attributes are based on the volumes in the pattern and not the current

settings. Scaling is done in the active coordinate system. Volumes in the pattern could have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Scale>Volumes

VMESH, *NV1*, *NV2*, *NINC*

Generates nodes and volume elements within volumes.

PREP7: Meshing

MP ME ST DY <> PR EM EH FL PP ED

NV1, *NV2*, *NINC*

Mesh volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and all selected volumes [**VSEL**] are meshed. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

Notes

Missing nodes required for the generated elements are created and assigned the lowest available numbers [**NUMSTR**]. During a batch run and if elements already exist, a mesh abort will write an alternative database file (**File.DBE**) for possible recovery.

Tetrahedral mesh expansion [**MOPT**,TETEXPND,*Value*] is supported for both the **VMESH** and **FVMESH** commands.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Free
Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>4 to 6 sided

---

VOFFST, *NAREA*, *DIST*, *KINC*

Generates a volume, offset from a given area.

PREP7: Volumes

MP ME ST DY <> PR EM EH FL PP ED

NAREA

Area from which generated volume is to be offset. If *NAREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

DIST

Distance normal to given area at which keypoints for generated volume are to be located. Positive normal is determined from the right-hand rule keypoint order.

KINC

Increment to be applied to the keypoint numbers between sets. If zero, keypoint numbers will be automatically assigned beginning with the lowest available value [**NUMSTR**].

Notes

Generates a volume (and its corresponding keypoints, lines, and areas) by offsetting from an area. The direction of the offset varies with the given area normal. End slopes of the generated lines remain the same as those of the given pattern.

If element attributes have been associated with the input area via the **AATT** command, the opposite area generated by the **VOFFST** operation will also have those attributes (i.e., the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the areas are meshed or belong to meshed volumes, a 3-D mesh can be extruded with this command. Note that the *NDIV* argument on the **ESIZE** command should be set before extruding the meshed areas.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>Along Normal

---

VOLUMES

Specifies "Volumes" as the subsequent status topic.

PREP7: Status

MP ME ST DY <> PR EM <> FL PP ED

Notes

This is a status [**STAT**] topic command. Status topic commands are generated by the GUI and appear in the log file (**Jobname.LOG**) if status is requested for some items by choosing **Utility Menu> List> Status**. This command will be immediately followed by a **STAT** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT** command should immediately follow this command.

Menu Paths

This command cannot be accessed from a menu.

VOVLAP, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Overlaps volumes.

PREP7: Booleans

MP ME ST DY <> PR EM EH FL PP ED

NV1, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Numbers of volumes to be operated on. If *NV1* = ALL, *NV2* to *NV9* are ignored and all selected volumes are used. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Overlaps volumes. Generates new volumes which encompass the geometry of all the input volumes. The new volumes are defined by the regions of intersection of the input volumes, and by the complementary (non-intersecting) regions. See the *ANSYS Modeling and Meshing Guide* for an illustration. This operation is only valid when the region of intersection is a volume. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Overlap>Volumes

---

VPLOT, *NV1*, *NV2*, *NINC*, *DEGEN*, *SCALE*

Displays the selected volumes.

PREP7: Volumes

MP ME ST DY <> PR EM <> FL PP ED

NV1, *NV2*, *NINC*
Display volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL (default), *NV2* and *NINC* are ignored and all selected volumes [**VSEL**] are displayed.

DEGEN
Degeneracy marker:

(blank)
No degeneracy marker is used (default).

DEGE
A red star is placed on keypoints at degeneracies (see the *ANSYS Modeling and Meshing Guide*). Not available if **/FACET**,WIRE is set.

SCALE
Scale factor for the size of the degeneracy-marker star. The scale is the size in window space (-1 to 1 in both directions) (defaults to .075).

Notes

Displays selected volumes. (Only volumes having areas within the selected area set [**ASEL**] will be plotted.) With PowerGraphics on [**/GRAPHICS**,POWER], **VPLOT** will display only the currently selected areas. This command is also a utility command, valid anywhere. The degree of tessellation used to plot the volumes is set through the **/FACET** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>Plot Degen Volus
Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>Plot Degen Volus
Utility Menu>Plot>Specified Entities>Volumes
Utility Menu>Plot>Volumes

VPTN, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Partitions volumes.

PREP7: Booleans

MP ME ST DY <> PR EM EH FL PP ED

NV1, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*
Numbers of volumes to be operated on. If *NV1* = ALL, *NV2* to *NV9* are ignored and all selected volumes are used. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Partitions volumes. Generates new volumes which encompass the geometry of all the input volumes. The new volumes are defined by the regions of intersection of the input volumes, and by the complementary (non-intersecting) regions. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Partition>Volumes

---

VPUT, *Par*, *IR*, *TSTRT*, *KCPLX*

Moves an array parameter vector into a variable.

POST26: Special Purpose

MP ME ST DY <> PR EM <> FL PP ED

Par
Array parameter vector in the operation.

IR
Arbitrary reference number assigned to this variable (1 to NV [**NUMVAR**]). Overwrites any existing results for this variable.

TSTRT
Time (or frequency) corresponding to start of *IR* data. If between values, the nearer value is used.

KCPLX
Complex number key:

0
Use the real part of the *IR* data.

1
Use the imaginary part of the *IR* data.

Notes

At least one variable should be defined (**NSOL**, **ESOL**, **RFORCE**, etc.) before using this command. The starting array element number must be defined. For example, **VPUT**,A(1),2 moves array parameter A to variable 2 starting at time 0.0. Looping continues from array element A(1) with the index number incremented by one until the

variable is filled. Unfilled variable locations are assigned a zero value. The number of loops may be controlled with the ***VLEN** command (except that loop skipping (NINC) is not allowed). For multi-dimensioned array parameters, only the first (row) subscript is incremented.

Menu Paths

Main Menu>TimeHist Postpro>Table Operations>Parameter to Var

VROTAT, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *PAX1*, *PAX2*, *ARC*, *NSEG*

Generates cylindrical volumes by rotating an area pattern about an axis.

PREP7: Volumes

MP ME ST DY <> PR EM EH FL PP ED

NA1, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*

List of areas in the pattern to be rotated (6 maximum if using keyboard entry). Areas must lie to one side of, and in the plane of, the axis of rotation. If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NA1* = ALL, all selected areas will define the pattern to be rotated. A component name may also be substituted for *NA1*.

PAX1, *PAX2*

Keypoints defining the axis about which the area pattern is to be rotated.

ARC

Arc length (in degrees). Positive follows right-hand rule about *PAX1-PAX2* vector. Defaults to 360.

NSEG

Number of volumes (8 maximum) around circumference. Defaults to minimum required for 90° (maximum) arcs, i.e., 4 for 360°, 3 for 270°, etc.

Notes

Generates cylindrical volumes (and their corresponding keypoints, lines, and areas) by rotating an area pattern (and its associated line and keypoint patterns) about an axis. Keypoint patterns are generated at regular angular locations (based on a maximum spacing of 90°). Line patterns are generated at the keypoint patterns. Arc lines are also generated to connect the keypoints circumferentially. Keypoint, line, area, and volume numbers are automatically assigned (beginning with the lowest available values). Adjacent lines use a common keypoint, adjacent areas use a common line, and adjacent volumes use a common area.

To generate a single volume with an arc greater than 180°, *NSEG* must be greater than or equal to 2.

If element attributes have been associated with the input area via the **AATT** command, the opposite area generated by the **VROTAT** operation will also have those attributes (i.e., the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the given areas are meshed or belong to meshed volumes, the 2-D mesh can be rotated (extruded) to a 3-D mesh. See the *ANSYS Modeling and Meshing Guide* for more information. Note that the *NDIV* argument on the **ESIZE** command should be set before extruding the meshed areas.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>About Axis

VSBA, *NV*, *NA*, *SEPO*, *KEEPV*, *KEEPA*

Subtracts areas from volumes.

PREP7: Booleans

MP ME ST DY <> PR EM EH FL PP ED

NV

Volume (or volumes, if picking is used) to be subtracted from. If ALL, use all selected volumes. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NV*.

NA

Area (or areas, if picking is used) to subtract. If ALL, use all selected areas. A component name may also be substituted for *NA*.

SEP0

Behavior of the touching boundary:

(blank)
The resulting volumes will share area(s) where they touch.

SEPO
The resulting volumes will have separate, but coincident area(s) where they touch.

KEEPV

Specifies whether *NV* volumes are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NV* volumes after **VSBA** operation (override **BOPTN** command settings).

KEEP
Keep *NV* volumes after **VSBA** operation (override **BOPTN** command settings).

KEEPA

Specifies whether *NA* areas are to be deleted:

(blank)
Use the setting of KEEP on the **BOPTN** command.

DELETE
Delete *NA* areas after **VSBA** operation (override **BOPTN** command settings).

KEEP
Keep *NA* areas after **VSBA** operation (override **BOPTN** command settings).

Notes

Generates new volumes by subtracting the regions common to both the volumes and areas (the intersection) from the *NV* volumes. The intersection will be an area(s). If *SEP0* is blank, the volume is divided at the area and the resulting volumes will be connected, sharing a common area where they touch. If *SEP0* is set to SEPO, the volume is divided into two unconnected volumes with separate areas where they touch. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Volume by Area
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Volume by Area

---

VSBV, *NV1*, *NV2*, *SEPO*, *KEEP1*, *KEEP2*

Subtracts volumes from volumes.

PREP7: Booleans

MP ME ST DY <> PR EM EH FL PP ED

NV1
Volume (or volumes, if picking is used) to be subtracted from. If ALL, use all selected volumes. Volumes specified in set *NV2* are removed from set *NV1*. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NV1*.

NV2
Volume (or volumes, if picking is used) to subtract. If ALL, use all selected volumes (except those included in the *NV1* argument). A component name may also be substituted for *NV2*.

SEP0
Behavior if the intersection of the *NV1* volumes and the *NV2* volumes is an area or areas:

(blank)
The resulting volumes will share area(s) where they touch.

SEPO
The resulting volumes will have separate, but coincident area(s) where they touch.

KEEP1
Specifies whether *NV1* volumes are to be deleted:

(blank)
Use the setting of *KEEP* on the **BOPTN** command.

DELETE
Delete *NV1* volumes after **VSBV** operation (override **BOPTN** command settings).

KEEP
Keep *NV1* volumes after **VSBV** operation (override **BOPTN** command settings).

KEEP2
Specifies whether *NV2* volumes are to be deleted:

(blank)
Use the setting of *KEEP* on the **BOPTN** command.

DELETE
Delete *NV2* volumes after **VSBV** operation (override **BOPTN** command settings).

KEEP
Keep *NV2* volumes after **VSBV** operation (override **BOPTN** command settings).

Notes

Generates new volumes by subtracting the regions common to both *NV1* and *NV2* volumes (the intersection) from the *NV1* volumes. The intersection can be an volume(s) or area(s). If the intersection is an area and *SEP0* is blank, the *NV1* volume is divided at the area and the resulting volumes will be connected, sharing a common area where they touch. If *SEP0* is set to SEPO, *NV1* is divided into two unconnected volumes with separate areas where they touch. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **VSBV**,ALL,ALL will have no effect because all the volumes in set *NV1*will have been moved to set *NV2*.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Volumes
Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>With Options>Volumes
Main Menu>Preprocessor>Modeling>Operate>Subract>Volumes

VSBW, *NV*, *SEPO*, *KEEP*

Subtracts intersection of the working plane from volumes (divides volumes).

PREP7: Booleans

MP ME ST DY <> PR EM EH FL PP ED

NV
Volume (or volumes, if picking is used) to be subtracted from. If *NV* = ALL, use all selected volumes. If *NV* = P, graphical picking is enabled (valid only in the GUI). A component name may also be input for *NV*.

SEP0
Behavior of the created boundary.

(blank)
The resulting volumes will share area(s) where they touch.

SEPO
The resulting volumes will have separate, but coincident area(s).

KEEP
Specifies whether *NV* volumes are to be deleted.

(blank)
Use the setting of *KEEP* on the **BOPTN** command.

DELETE
Delete *NV* volumes after **VSBW** operation (override **BOPTN** command settings).

KEEP
Keep *NV* volumes after **VSBW** operation (override **BOPTN** command settings).

Notes

Generates new volumes by subtracting the intersection of the working plane from the *NV* volumes. The intersection will be an area(s). If *SEP0* is blank, the volume is divided at the area and the resulting volumes will be connected, sharing a common area where they touch. If *SEP0* is set to SEPO, the volume is divided into two unconnected volumes with separate areas. The SEPO option may cause unintended consequences if any keypoints exist along the cut plane. See the *ANSYS Modeling and Meshing Guide* for an illustration. See the **BOPTN** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Issuing the **VSBW** command under certain conditions may generate a topological degeneracy error. Do not issue the command if:

- A sphere or cylinder has been scaled. (A cylinder must be scaled unevenly in the XY plane.)
- A sphere or cylinder has not been scaled but the work plane has been rotated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Volu by WrkPlane
Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Volu by WrkPlane
Main Menu>Preprocessor>Modeling>Operate>Divide>Volu by WrkPlane

---

/VSCALE, *WN*, *VRATIO*, *KEY*

Scales the length of displayed vectors.

GRAPHICS: Scaling

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

VRATIO
Ratio value applied to the automatically calculated scale factor (defaults to 1.0, i.e., use scale factor as automatically calculated).

KEY
Relative scaling key:

0
Use relative length scaling among vectors based on magnitudes.

1
Use uniform length scaling for all vector lengths.

Command Default

Automatic vector scaling.

Notes

Allows scaling of the vector length displayed with the **PLVECT** command of POST1 and the **/PBC** and **/PSF** commands. Also allows the scaling of the element (i.e., **/PSYMB**,ESYS) and the nodal (i.e., **/PSYMB**,NDIR) coordinate system symbols.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Plot Results>Vector Plot>Predefined
Utility Menu>Plot>Results>Vector Plot
Utility Menu>PlotCtrls>Multi-Plot Contrls
Utility Menu>PlotCtrls>Style>Vector Arrow Scaling

VSEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KSWP*

Selects a subset of volumes.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type
Label identifying the type of volume select:

S
Select a new set (default).

R
Reselect a set from the current set.

A
Additionally select a set and extend the current set.

U
Unselect a set from the current set.

ALL
Restore the full set.

NONE
Unselect the full set.

INVE
Invert the current set (selected becomes unselected and vice versa).

STAT
Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Item
Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to VOLU.

Comp
Component of the item (if required). Valid component labels are shown in the table below.

VMIN
Minimum value of item range. Ranges are volume numbers, coordinate values, attribute numbers, etc., as appropriate for the item. A component name (as specified on the **CM** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, or ESYS and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **VLIST** command for a discussion of signed attributes.

VMAX
Maximum value of item range. *VMAX* defaults to *VMIN*. If *VMAX* = *VMIN* , a tolerance of ±0.005x *VMIN* is used, or ±1.0E-6 if *VMIN* = 0.0. If *VMAX* ≠ *VMIN*, a tolerance of ±1.0E-8x (*VMAX*-*VMIN*) is used.

VINC
Value increment within range. Used only with integer ranges (such as for volume numbers). Defaults to 1. *VINC* cannot be negative.

KSWP
Specifies whether only volumes are to be selected:

0
Select volumes only.

1
Select volumes, as well as keypoints, lines, areas, nodes, and elements associated with selected volumes. Valid only with *Type* = S.

Command Default

All volumes are selected.

Notes

Selects volumes based on values of a labeled item and component. For example, to select a new set of volumes based on volume numbers 1 through 7, use **VSEL**,S,VOLU,,1,7. The subset is used when the ALL label is entered (or implied) on other commands, such as **VLIST**,ALL. Only data identified by volume number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

This command is valid in any processor.

VSEL - Valid Item and Component Labels

VSEL*Type, Item, Comp, VMIN, VMAX, VINC, KABS*

Item	Comp	Description
VOLU		Volume number.
LOC	X, Y, Z	X, Y, or Z center (picking "hot spot" location in the active coordinate system).
MAT		Material number associated with the volume.
TYPE		Element type number associated with the volume.
REAL		Real constant set number associated with the volume.
ESYS		Element coordinate system associated with the volume.

Menu Paths

Utility Menu>Select>Entities

VSLA, *Type*, *VLKEY*

Selects those volumes containing the selected areas.

DATABASE: Selecting

MP ME ST DY <> PR EM <> FL PP ED

Type

Label identifying the type of volume select:

S
: Select a new set (default).

R
: Reselect a set from the current set.

A
: Additionally select a set and extend the current set.

U
: Unselect a set from the current set.

VLKEY

Specifies whether all contained volume areas must be selected [**ASEL**]:

0
: Select volume if any of its areas are in the selected area set.

1
: Select volume only if all of its areas are in the selected area set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

VSUM, *LAB*

Calculates and prints geometry statistics of the selected volumes.

PREP7: Volumes

MP ME ST DY <> PR EM <> FL PP ED

LAB

Controls the degree of tessellation used in the calculation of area properties. If *LAB* = DEFAULT, area calculations will use the degree of tessellation set through the **/FACET** command. If *LAB* = FINE, area calculations are based on a finer tessellation.

Notes

Calculates and prints geometry statistics (volume, centroid location, moments of inertia, etc.) associated with the selected volumes. Geometry items are reported in the global Cartesian coordinate system. A unit density is assumed unless the volumes have a material association via the **VATT** command. Items calculated by **VSUM** and later retrieved by a ***GET** or ***VGET** command are valid only if the model is not modified after the **VSUM** command is issued.

Setting a finer degree of tessellation will provide area calculations with greater accuracy, especially for thin, hollow models. However, using a finer degree of tessellation requires longer processing.

For very thin volumes, such that the ratio of the minimum to the maximum dimension is less than 0.01, the **VSUM** command can provide erroneous volume information. To ensure that such calculations are accurate, make certain that you subdivide such volumes so that the ratio of the minimum to the maximum is at least 0.05.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Volumes

---

VSWEEP, *VNUM*, *SRCA*, *TRGA*, *LSMO*

Fills an existing unmeshed volume with elements by sweeping the mesh from an adjacent area through the volume.

PREP7: Meshing

MP ME ST DY <> PR EM <> FL PP ED

VNUM

Number identifying the volume that is to be meshed by **VSWEEP**. If *VNUM* = P, graphical picking is enabled, you will be prompted to choose the volume or volumes based on the setting of **EXTOPT**,VSWE,AUTO. This argument is required.

"ALL" is a valid input value that when selected sends all the selected volumes to the sweeper. If *VNUM* = ALL, each volume that can be swept will be and those not able to be swept will be unmeshed or meshed with tets depending upon the setting of **EXTOPT**,VSWE,TETS.

A component name is a valid input value. All volumes that are part of the specified component will be sent to the sweeper.

SRCA

Number identifying the source area. This is the area whose mesh will provide the pattern for the volume elements. (If you do not mesh the source area prior to volume sweeping, ANSYS meshes it internally when you initiate volume sweeping.) ANSYS sweeps the pattern of the area elements through the volume to create the mesh of volume elements. You cannot substitute a component name for *SRCA*.

This argument is optional. If *VNUM* = ALL or is a component containing more than one volume, *SRCA* is ignored. If *SRCA* is not provided or if it is ignored, **VSWEEP** attempts to automatically determine which area should be the target area.

TRGA

Number identifying the target area. This is the area that is opposite the source area specified by *SRCA*. You cannot substitute a component name for *TRGA*.

This argument is optional. If *VNUM* = ALL or component containing more than one volume, *TRGA* is ignored. If *TRGA* is not provided or if it is ignored, **VSWEEP** attempts to automatically determine which area should be the target area.

LSMO
Value specifying whether ANSYS should perform line smoothing during volume sweeping. (The value of this argument controls line smoothing for the **VSWEEP** command only; it has no effect on the setting of the **MOPT** command's LSMO option.) This argument is optional.

0
Do not perform line smoothing. This is the default.

1
Always perform line smoothing. This setting is not recommended for large models due to speed considerations.

Notes

If the source mesh consists of quadrilateral elements, ANSYS fills the volume with hexahedral elements. If the source mesh consists of triangles, ANSYS fills the volume with wedges. If the source mesh consists of a combination of quadrilaterals and triangles, ANSYS fills the volume with a combination of hexahedral and wedge elements.

In the past, you may have used the **VROTAT**, **VEXT**, **VOFFST**, and/or **VDRAG** commands to extrude a meshed area into a meshed volume. However, those commands create the volume and the volume mesh simultaneously. In contrast, the **VSWEEP** command is intended for use in an existing unmeshed volume. This makes **VSWEEP** particularly useful when you have imported a solid model that was created in another program, and you want to mesh it in ANSYS.

For related information, see the description of the **EXTOPT** command, as well as the detailed discussion of volume sweeping in Meshing Your Solid Model of the *ANSYS Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volume Sweep>Sweep

VSYMM, *Ncomp*, *NV1*, *NV2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Generates volumes from a volume pattern by symmetry reflection.

PREP7: Volumes

MP ME ST DY <> PR EM EH FL PP ED

Ncomp
Symmetry key:

X
X symmetry (default).

Y
Y symmetry.

Z
Z symmetry.

NV1, *NV2*, *NINC*
Reflect volumes from pattern beginning with *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is all selected volumes [**VSEL**]. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

KINC
Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM
Specifies whether nodes and elements are also to be generated:

0
Generate nodes and elements associated with the original volumes, if they exist.

1
Do not generate nodes and elements.

IMOVE
Specifies whether volumes will be moved or newly defined:

0
Generate additional volumes.

1
Move original volumes to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a reflected set of volumes (and their corresponding keypoints, lines, areas and mesh) from a given volume pattern by a symmetry reflection (see analogous node symmetry command, **NSYM**). The MAT, TYPE, REAL, and ESYS attributes are based upon the volumes in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. The active coordinate system must be a Cartesian system. Volumes in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Volumes are generated as described in the **VGEN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Volumes

VTRAN, *KCNTO*, *NV1*, *NV2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Transfers a pattern of volumes to another coordinate system.

PREP7: Volumes

MP ME ST DY <> PR EM EH FL PP ED

KCNTO
Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system. The coordinate system type and parameters of *KCNTO* must be the same as the active system.

NV1, *NV2*, *NINC*
Transfer volumes from pattern beginning with *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is all selected volumes [**VSEL**]. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

KINC
Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned [**NUMSTR**].

NOELEM
Specifies whether elements and nodes are also to be generated:

0
Generate nodes and elements associated with the original volumes, if they exist.

1
Do not generate nodes and elements.

IMOVE
Specifies whether to redefine the existing volumes:

0
Generate additional volumes.

1
Move original volumes to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Corresponding meshed items are also moved if not needed at their original position.

Notes

Transfers a pattern of volumes (and their corresponding keypoints, lines, areas and mesh) from one coordinate system to another (see analogous node transfer command, **TRANSFER**). The MAT, TYPE, REAL, and ESYS attributes are based upon the volumes in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Coordinate and slope values are interpreted in the active coordinate system and are transferred directly. Volumes are generated as described in the **VGEN** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Volumes

---

VTYPE, *NOHID*, *NZONE*

Specifies the viewing procedure used to determine the form factors for the Radiation Matrix method.

AUX12: Radiation Substructures

MP ME ST <> <> PR <> <> <> PP ED

NOHID
Type of viewing procedure:

0
Hidden procedure.

1
Non-hidden (faster, but less general) procedure.

NZONE
Number of sampling zones for the hidden procedure (100 maximum for 3-D, 1000 maximum for 2-D). Defaults to 20 for 3-D, 200 for 2-D. Number of points is 2**NZONE* for 2-D and 2**NZONE**(*NZONE*+1) for 3-D.

Command Default

Hidden procedure using 20 zones for 3-D, 200 zones for 2-D.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Write Matrix

/VUP, *WN*, *Label*
Specifies the global Cartesian coordinate system reference orientation.

GRAPHICS: Views

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number (or ALL) to which command applies (defaults to 1).

Label
Orientation:

Y
Y vertical upward, X horizontal to the right, Z out from the screen (default).

Y
Y vertical downward, X horizontal to the left, Z out from the screen.

X
X vertical upward, Y horizontal to the left, Z out from the screen.

X
X vertical downward, Y horizontal to the right, Z out from the screen.

Z
Z vertical upward, Y horizontal to the right, X out from the screen. With this choice, you should use a view other than the **/VIEW** default of (0,0,1).

Z
Z vertical downward, Y horizontal to the left, X out from the screen. With this choice, you should use a view other than the **/VIEW** default of (0,0,1).

Command Default

Y vertical upward, X horizontal to the right, Z out from the screen.

Notes

Specifies the global Cartesian coordinate system reference orientation. The **/VIEW** and **/ANGLE** commands may be used to reorient the view and are relative to this reference orientation. All coordinate systems are right-handed.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>View Settings>Viewing Direction

W Commands

WAVES, *Wopt*, *OLDMAX*, *OLDRMS*

Initiates reordering.

PREP7: Element Reordering

MP ME ST <> <> PR EM <> <> PP ED

Wopt

Option for comparison:

MAX

Use maximum wavefront value for comparison (default).

RMS

Use RMS wavefront value for comparison.

OLDMAX, *OLDRMS*

Maximum and RMS wavefront values of model to be used in place of the old values. *OLDRMS* defaults to *OLDMAX* (and vice versa). If neither is specified, each defaults to its calculated old value.

Notes

Initiates the element reordering based upon the previously defined starting wave lists (if any). Reordering affects only the element order for the solution phase and not the element numbers (input referring to element numbers, such as element pressures, is unaffected by reordering).

Note — The new order is retained only if new the new maximum or RMS wavefront values are lower than the old values, as described below. See the **WSORT** command for another reordering procedure. The resulting element ordering can be shown by listing the wavefront history [**WFRONT**,1] or by displaying elements with their element location numbers [**/PNUM**].

The **WAVES** reordering procedure is based upon a list of nodes defining where the element reordering is to start (either input on the **WSTART** command, or automatically determined if not input). If a list is input, additional starting wave lists may be defined to allow the user to block or guide the wave. An input starting list usually consists of one node for a line element model, a line of nodes for an area element model, or a plane of nodes for a volume element model. Elements are ordered from the first starting wave list in a direction that attempts to minimize the wavefront. Nodes are deleted and added to the total wave list set as reordering progresses through the model. Elements attached to nodes on succeeding starting lists, if any, are ignored at this time, thereby temporarily blocking that path. Whenever no more elements can be reordered, the procedure is repeated starting from the nodes defined on the second starting list (if any). The same node number should not appear on more than one list. If no starting list is input, a starting node is automatically selected from an element weighting procedure (see the *ANSYS, Inc. Theory Reference*).

Reordering may be made from the previously reordered model or from the initial model (by issuing the **SAVE** command before reordering and then restoring that model back to PREP7 with the **RESUME** command after reordering). The **WFRONT** command is useful for calculating and printing the current wavefront statistics at any time without causing any reordering of the elements. If a model is to be used for both a thermal and a structural analysis, the reordered statistics should be based upon the structural model (because of the higher number of degrees of freedom per node).

The reordering procedure treats separate portions of the model (i.e., not connected by elements) as discontinuous. Reordering automatically continues across a discontinuity as best as possible.

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Element Reorder>Reorder by List

---

WERASE

Erases all reordering wave lists.

PREP7: Element Reordering

MP ME ST <> <> PR EM <> <> PP ED

Notes

The **REORDER** then **STAT** commands will display the current wave lists.

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Element Reorder>Erase Wave List

---

WFRONT, *KPRNT*, *KCALC*

Estimates wavefront statistics.

PREP7: Element Reordering

MP ME ST <> <> PR EM <> <> PP ED

KPRNT

Wavefront history print key:

0

Print current wavefront statistics.

1

Print current wavefront statistics but also print wavefront history (wavefront at each element). Elements are listed in the reordered sequence.

KCALC

Calculation options:

0

Wavefront estimate assumes maximum model DOF set at each node and does not include the effects of master degrees of freedom and specified displacement constraints.

1

Wavefront estimate uses the actual DOF set at each node and does not include the effects of master degrees of freedom and specified displacement constraints. More time consuming than estimated wavefront. *KPRNT* = 1 is not available with this option.

Notes

Estimates wavefront statistics of the model as currently ordered.

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Element Reorder>Est. Wavefront

---

/WINDOW, *WN*, *XMIN*, *XMAX*, *YMIN*, *YMAX*, *NCOPY*

Defines the window size on the screen.

GRAPHICS: Set Up

MP ME ST DY <> PR EM <> FL PP ED

WN

Window reference number (1 to 5). Defaults to 1. This number, or ALL (for all active windows), may be used on other commands.

XMIN, *XMAX*, *YMIN*, *YMAX*

Screen coordinates defining window size. Screen coordinates are measured as -1.0 to 1.67 with the origin at the screen center. For example, (-1,1.67,-1,1) is full screen, (-1,0,-1,0) is the left bottom quadrant. If *XMIN* = OFF, deactivate this previously defined window; if ON, reactivate this previously defined window. If FULL, LEFT, RIGH, TOP, BOT, LTOP, LBOT, RTOP, RBOT, form full, half, or quarter window. If SQUA, form largest square window within the current graphics area. If DELE, delete this window (cannot be reactivated with ON).

NCOPY

Copies the current specifications from window *NCOPY* (1 to 5) to this window. If *NCOPY* = 0 (or blank), no specifications are copied.

Command Default

One window at full screen.

Notes

Defines the window size on the screen. Windows may occupy a separate section of the screen or they may overlap. Requested displays are formed in all windows according to the selected window specifications.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Window Controls>Copy Window Specs
Utility Menu>PlotCtrls>Window Controls>Delete Window
Utility Menu>PlotCtrls>Window Controls>Window Layout
Utility Menu>PlotCtrls>Window Controls>Window On or Off

WMID, *Key*

Specifies reordering options for the WAVES command.

PREP7: Element Reordering

MP ME ST <> <> PR EM <> <> PP ED

Key

Determines whether midside nodes are considered when reordering.

NO

Do not consider midside nodes when reordering (default).

YES

Consider midside nodes when reordering. This option is useful for models where line elements are only attached to midside nodes of solid elements.

Menu Paths

This command cannot be accessed from a menu.

WMORE, *NODE1*, *NODE2*, *NINC*, *ITIME*, *INC*

Adds more nodes to the starting wave list.

PREP7: Element Reordering

MP ME ST <> <> PR EM <> <> PP ED

NODE1, *NODE2*, *NINC*

Add another node set to the previous starting wave list. Set is *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* is negative, delete (instead of add) this node set from previous starting wave list.

ITIME, *INC*

Add other node sets to the same starting wave list by repeating the previous node set with *NODE1* and *NODE2* incremented by *INC* (defaults to 1) each time after the first. *ITIME* is the total number of sets (defaults to 1) defined with this command.

Notes

Adds more nodes to (or modifies) the previous starting wave list (if any) [**WSTART**]. Repeat **WMORE** command to add more nodes to the previous starting wave list. Up to 10,000 nodes may be defined (total, for all starting waves).

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Element Reorder>Extend Wave List

---

WPAVE, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*, *X3*, *Y3*, *Z3*

Moves the working plane origin to the average of specified points.

DATABASE: Working Plane

MP ME ST DY <> PR EM <> FL PP ED

X1, *Y1*, *Z1*
Coordinates (in the active coordinate system) of the first point. If *X1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

X2, *Y2*, *Z2*
Coordinates (in the active coordinate system) of the second point.

X3, *Y3*, *Z3*
Coordinates (in the active coordinate system) of the third point.

Notes

Moves the origin of the working plane to the average of the specified points. A point is considered specified only if at least one of its coordinates is non-blank, and at least one point (1, 2, or 3) must be specified. Blank coordinates of a specified point are assumed to be zero. Averaging is based on the active coordinate system.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Offset WP to>Global Origin
Utility Menu>WorkPlane>Offset WP to>Origin of Active CS
Utility Menu>WorkPlane>Offset WP to>XYZ Locations

WPCSYS, *WN*, *KCN*

Defines the working plane location based on a coordinate system.

DATABASE: Working Plane

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified.

KCN
Coordinate system number. *KCN* may be 0,1,2 or any previously defined local coordinate system number (defaults to the active system).

Notes

Defines a working plane location and orientation based on an existing coordinate system. If a Cartesian system is used as the basis (*KCN*) for the working plane, the working plane will also be Cartesian, in the X-Y plane of the base system. If a cylindrical, spherical, or toroidal base system is used, the working plane will be a polar system in the R-θ plane of the base system.

If working plane tracking has been activated (**CSYS**,WP or **CSYS**,4), the updated active coordinate system will be of a similar type, except that a toroidal system will be updated to a cylindrical system. See the *ANSYS Modeling and Meshing Guide* for more information on working plane tracking.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node
Utility Menu>WorkPlane>Align WP with>Active Coord Sys
Utility Menu>WorkPlane>Align WP with>Global Cartesian
Utility Menu>WorkPlane>Align WP with>Specified Coord Sys

WPLANE, *WN*, *XORIG*, *YORIG*, *ZORIG*, *XXAX*, *YXAX*, *ZXAX*, *XPLAN*, *YPLAN*, *ZPLAN*

Defines a working plane to assist in picking operations.

DATABASE: Working Plane

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified. If fewer than three points are used, the viewing direction of window *WN* will be used instead to define the normal to the working plane.

XORIG, *YORIG*, *ZORIG*
Global Cartesian coordinates of the origin of the working plane coordinate system.

XXAX, *YXAX*, *ZXAX*
Global Cartesian coordinates of a point defining the x-axis orientation. The x-axis aligns with the projection of the line from this orientation point to the origin.

XPLAN, *YPLAN*, *ZPLAN*
Global Cartesian coordinates of the third point defining the working plane. This point will also define the location of the positive XY-sector of the working plane coordinate system.

Command Default

Working plane parallel to the global X-Y plane at Z = 0.0.

Notes

Defines a working plane to assist in picking operations using the coordinates of three noncolinear points. The three points also define the working plane coordinate system. A minimum of one point (the working plane origin) is required. Immediate mode may also be active. See **WPSTYL** command to set the style of working plane display.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Align WP with>XYZ Locations

WPOFFS, *XOFF*, *YOFF*, *ZOFF*

Offsets the working plane.

DATABASE: Working Plane

MP ME ST DY <> PR EM <> FL PP ED

X0FF, *Y0FF*, *Z0FF*

Offset increments defined in the working plane coordinate system. If only *Z0FF* is used, the working plane will be redefined parallel to the present plane and offset by *Z0FF*.

Notes

Changes the origin of the working plane by translating the working plane along its coordinate system axes.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Offset WP by Increments

WPROTA, *THXY*, *THYZ*, *THZX*

Rotates the working plane.

DATABASE: Working Plane

MP ME ST DY <> PR EM <> FL PP ED

THXY

First rotation about the working plane Z axis (positive X toward Y).

THYZ

Second rotation about working plane X axis (positive Y toward Z).

THZX

Third rotation about working plane Y axis (positive Z toward X).

Notes

The specified angles (in degrees) are relative to the orientation of the working plane.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Offset WP by Increments

WPSTYL, *SNAP*, *GRSPAC*, *GRMIN*, *GRMAX*, *WPTOL*, *WPCTYP*, *GRTYPE*, *WPVIS*, *SNAPANG*

Controls the display and style of the working plane.

DATABASE: Working Plane

MP ME ST DY <> PR EM <> FL PP ED

SNAP

Snap increment for a locational pick (1E-6 minimum). If -1, turn off snap capability. For example, a picked location of 1.2456 with a snap of 0.1 gives 1.2, with 0.01 gives 1.25, with 0.001 gives 1.246, and with 0.025 gives 1.250 (defaults to 0.05).

GRSPAC

Graphical spacing between grid points. For graphical representation only and not related to snap points (defaults to 0.1).

GRMIN, *GRMAX*

Defines the size of a square grid (if *WPCTYP* = 0) to be displayed over a portion of the working plane. The opposite corners of the grid will be located at grid points nearest the working plane coordinates of (*GRMIN*,*GRMIN*) and (*GRMAX*,*GRMAX*). If a polar system (*WPCTYP* = 1), *GRMAX* is the outside radius of grid and *GRMIN* is ignored. If *GRMIN* = *GRMAX*, no grid will be displayed (defaults to -1.0 and 1.0 for *GRMIN* and *GRMAX* respectively).

WPTOL

The tolerance that an entity's location can deviate from the specified working plane, while still being considered on the plane. Used only for locational picking of vertices for polygons and prisms (defaults to 0.003).

WPCTYP

Working plane coordinate system type:

0

Cartesian (default). If working plane tracking is on [**CSYS**,4], the updated active coordinate system will also be Cartesian.

1

Polar. If working plane tracking is on, the updated active coordinate system will be cylindrical.

2

Polar. If working plane tracking is on, the updated active coordinate system will be spherical.

GRTYPE

Grid type:

0

Grid and WP triad.

1

Grid only.

2

WP triad only (default).

WPVIS

Grid visibility:

0

Do not show *GRTYPE* entities (grid and/or triad) (default).

1
Show *GRTYPE* entities. Cartesian working planes will be displayed with a Cartesian grid, polar with a polar grid.

SNAPANG
Snap angle (0--180) in degrees. Used only if *WPCTYP* = 1 or 2. Defaults to 5 degrees.

Command Default

Snap capability on with an increment of .05, spacing between grid points of 0.1, Cartesian coordinate system, WP triad displayed, and a tolerance of .003.

Notes

Use **WPSTYL**,DEFA to reset the working plane to its default location and style. Use **WPSTYL**,STAT to list the status of the working plane. Blank fields will keep present settings.

It is possible to specify *SNAP* and *WPTOL* values that will cause conflicts during picking operations. Check your values carefully, and if problems are noted, revert to the default values.

WPSTYL with no arguments will toggle the grid on and off. The working plane can be displayed in the non-GUI interactive mode only after issuing a **/PLOPTS**,WP,1 command. See the *ANSYS Modeling and Meshing Guide* for more information on working plane tracking. See **/PLOPTS** command for control of hidden line working plane.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>ROM>ElecStruc
Main Menu>Preprocessor>Modeling>Create>Circuit>Set Grid
Utility Menu>List>Status>Working Plane
Utility Menu>WorkPlane>Display Working Plane
Utility Menu>WorkPlane>Offset WP by Increments
Utility Menu>WorkPlane>Show WP Status
Utility Menu>WorkPlane>WP settings

WRFULL, *Ldstep*

Stops solution after assembling global matrices.

SOLUTION: Load Step Options

MP ME ST <> <> PR EM <> <> PP ED

Ldstep
Specify action to take:

OFF or 0
Turn off feature (default)

N
Turn on feature and set it to stop after assembling the global matrices and writing the **.FULL** file for load step N.

Command Default

By default the WRFULL command is turned OFF.

Notes

This command is used in conjunction with the **SOLVE** command to generate the assembled matrix file (**.FULL** file) only. The element matrices are assembled into the relevant global matrices for the particular analysis being performed and the **.FULL** file is written. Equation solution and the output of data to the results file are skipped. To dump the matrices written on the **.FULL** file into Harwell-Boeing format, use the HBMAT command in /AUX2.

To use the **LSSOLVE** macro with this command, you may need to modify the **LSSOLVE** macro to properly stop at the load step of interest.

This command only valid for linear static, full harmonic, and full transient analyses when the sparse direct solver is selected. This command is also valid for bucking or modal analyses with any mode extraction method. This command is not valid for nonlinear analyses or analyses containing p-elements.

In general, the assembled matrix file **.FULL** contains stiffness, mass, and damping matrices. However, the availability of the matrices depends on the analysis type chosen when the file is written.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Stop Solution
Main Menu>Solution>Load Step Opts>Stop Solution

WRITE, *Fname*

Writes the radiation matrix file.

AUX12: Radiation Substructures

MP ME ST <> <> PR <> <> <> PP ED

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name Defaults to **Jobname**.

Notes

Writes radiation matrix file (**File.SUB**) for input to the substructure thermal "use" pass. Subsequent **WRITE** operations to the same file overwrite the file.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Write Matrix

WSORT, *Lab*, *KORD*, --, *Wopt*, *OLDMAX*, *OLDRMS*

Initiates element reordering based upon a geometric sort.

PREP7: Element Reordering

MP ME ST <> <> PR EM <> <> PP ED

Lab
Coordinate (in the active system) along which element centroid locations are sorted. Valid labels are: X, Y, Z, ALL. If ALL (default), all three directions will be used, and the order corresponding to the lowest MAX or RMS wavefront value will be retained.

KORD
Sort order:

0
Sort according to ascending coordinate values.

1
Sort according to descending coordinate values.

--
Unused field.

Wopt
Option for comparison:

MAX
Use maximum wavefront value for comparison (default).

RMS
Use RMS wavefront value.

OLDMAX, *OLDRMS*
MAX and RMS wavefront values of model to be used in place of the old values. *OLDRMS* defaults to *OLDMAX* (and vice versa). If neither is specified, each defaults to its calculated old value.

Notes

Initiates element reordering based upon a geometric sort of the element centroid locations. Wave lists, if any [**WSTART**], are ignored. Reordering affects only the element order for the solution phase and not the element numbers (input referring to element numbers, such as element pressures, is unaffected by reordering).

Note — The new order is retained only if new the new maximum or RMS wavefront values are lower than the old values, as described below. See the **WAVES** command for another reordering procedure and for more details on reordering. The resulting element ordering can be shown by listing the wavefront history [**WFRONT**,1] or by displaying elements with their element location numbers [**/PNUM**].

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Element Reorder>Reorder by XYZ

WSPRINGS

Creates weak springs on corner nodes of a bounding box of the currently selected elements.

SOLUTION: Misc Loads

MP ME ST <> <> <> <> <> <> <> ED

Notes

WSPRINGS invokes a predefined ANSYS macro that is used during the import of loads from the ADAMS program into the ANSYS program. **WSPRINGS** creates weak springs on the corner nodes of the bounding box of the currently selected elements. The six nodes of the bounding box are attached to ground using COMBIN14 elements. The stiffness is chosen as a small number and can be changed by changing the real constants of the COMBIN14 elements. This command works only for models that have a geometric extension in two or three dimensions. One dimensional problems (pure beam in one axis) are not supported.

For more information on how **WSPRINGS** is used during the transfer of loads from the ADAMS program to ANSYS, see Import Loads into ANSYS in the *ANSYS Advanced Analysis Techniques Guide*.

Menu Paths

Main Menu>Solution>ADAMS Connection>Import fr ADAMS

WSTART, *NODE1*, *NODE2*, *NINC*, *ITIME*, *INC*

Defines a starting wave list.

PREP7: Element Reordering

MP ME ST <> <> PR EM <> <> PP ED

NODE1, *NODE2*, *NINC*
: Define a set of nodes in the starting wave list from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, ignore remaining fields and use all selected nodes [**NSEL**].

ITIME, *INC*
: Add more node sets to the same starting wave list by repeating the previous node set with *NODE1* and *NODE2* incremented by *INC* (defaults to 1) each time after the first. *ITIME* is the total number of sets (defaults to 1) defined with this command.

Command Default

If no starting waves, a starting node will be automatically defined when the **WAVES** command is issued.

Notes

Defines a starting wave list (optional) for reordering with the **WAVES** command. Repeat **WSTART** command to define *other* starting wave lists (20 maximum).

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Element Reorder>Define Wave List

---

X Commands

/XFRM, *LAB*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

Controls the centroid or the axis of dynamic rotation.

GRAPHICS: Views

MP ME ST DY <> PR EM <> FL PP ED

LAB

The location or entity (centroid) used to define the center or axis of rotation.

NODE

If NODE is chosen for the center of rotation, the node number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two nodes between which a line is drawn to determine the axis. The remaining arguments are ignored.

ELEMENT

If ELEMENT is chosen for the center of rotation, the element number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two elements between which a line is drawn to determine the axis. The remaining arguments are ignored.

KP

If KP is chosen for the center of rotation, the keypoint number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two keypoints between which a line is drawn to determine the axis. The remaining arguments are ignored.

LINE

If LINE is chosen for the center of rotation, the line number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two lines between which a line is drawn to determine the axis. The remaining arguments are ignored.

AREA

If AREA is chosen for the center of rotation, the area number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two areas between which a line is drawn to determine the axis. The remaining arguments are ignored.

VOLUME

If VOLUME is chosen for the center of rotation, the volume number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two volumes between which a line is drawn to determine the axis. The remaining arguments are ignored.

XYZ

If XYZ is chosen for the center of rotation, the location of that center is determined by the coordinates *X1*, *Y1*, *Z1*. If values are specified for *X2*, *Y2*, *Z2*, then the axis of rotation will be about the line between those two points.

OFF

If *LAB* = OFF, DEFAULT, FOCUS or if no value is specified, then the center of rotation is set at the FOCUS point, as defined by the **/FOCUS** command.

X1

The entity number or X coordinate for the center of rotation.

Y1

The entity number or Y coordinate for the center of rotation.

Z1
The Z coordinate for the center of rotation.

X2
The X coordinate for the axis of rotation.

Y2
The Y coordinate for the axis of rotation.

Z2
The Z coordinate for the axis of rotation.

Command Default

Issuing **/XFRM**, with no *LAB* defined sets the center of rotation at the focal point specified by the **/FOCUS** command.

Notes

The **/XFRM** command is active only when the cumulative rotation key is specified ON for the **/ANGLE** command (*KINCR* = 1). This command affects dynamic manipulations only.

For center rotation, the middle mouse button will rotate the model about the screen Z axis and the right mouse button will rotate the model about the screen X and Y axis.

For rotation about an axis, the middle mouse button will rotate the model about the defined axis of rotation and the right mouse button will be deactivated.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>ViewSettings>Rotational Center>ByPick

/XRANGE, *XMIN*, *XMAX*

Specifies a linear abscissa (X) scale range.

GRAPHICS: Graphs

MP ME ST DY <> PR EM <> FL PP ED

XMIN
Minimum abscissa scale value.

XMAX
Maximum abscissa scale value.

Command Default

Automatically select X-range scale to include all data being displayed.

Notes

Specifies a linear abscissa (X) scale range for the line graph display. Use **/XRANGE**,DEFAULT to return to automatic scaling.

Automatic scaling will often yield inappropriate range values for logarithmic scales (**/GROPT**, LOGX).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

XVAR, *N*

Specifies the X variable to be displayed.

POST26: Display

MP ME ST DY <> PR EM <> FL PP ED

N

X variable number:

0 or 1
: Display **PLVAR** values vs. time (or frequency).

n
: Display **PLVAR** values vs. variable *n* (2 to *NV* [**NUMVAR**]).

1
: Interchange time and **PLVAR** variable numbers with time as the curve parameter. **PLVAR** variable numbers are displayed uniformly spaced along X-axis from position 1 to 10.

Command Default

Use time or frequency.

Notes

Defines the X variable (displayed along the abscissa) against which the Y variable(s) [**PLVAR**] are to be displayed.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph

XVAROPT, *Lab*

Specifies the parameter to be used as the X-axis variable.

OPTIMIZATION: Display

MP ME ST DY <> PR EM <> FL PP ED

Lab

Parameter to be used as the X variable. Defaults to set numbers.

Command Default

Use design set numbers as the X variable.

Notes

Specifies the parameter to be used as the X-axis variable for graphical displays produced by the **PLVAROPT** command, and for the first column of the tabular listings produced by the **PRVAROPT** command. Design sets are automatically sorted in a sequence corresponding to an ascending order of the specified parameter.

Menu Paths

Main Menu>Design Opt>Design Sets>Graphs/Tables

Y Commands

/YRANGE, *YMIN*, *YMAX*, *NUM*

Specifies a linear ordinate (Y) scale range.

GRAPHICS: Graphs

MP ME ST DY <> PR EM <> FL PP ED

YMIN
Minimum ordinate scale value.

YMAX
Maximum ordinate scale value.

NUM
Y-axis number to which range applies (defaults to 1). Valid numbers are 1 to 3 for **/GRTYP**,2 and 1 to 6 for **/GRTYP**,3. If ALL, apply to all Y-axes.

Command Default

Automatically select Y-range scale to include all data being displayed.

Notes

Specifies a linear ordinate (Y) scale range for the line graph display. Use **/YRANGE**,DEFAULT to return to automatic scaling. For multiple Y-axes graphs [**/GRTYP**], see **/GROPT**, ASCAL to automatically scale the additional Y-axes.

Automatic scaling will often yield inappropriate range values for logarithmic scales (**/GROPT**, LOGY).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

Z Commands

/ZOOM, *WN*, *Lab*, *X1*, *Y1*, *X2*, *Y2*
Zooms a region of a display window.

GRAPHICS: Views

MP ME ST DY <> PR EM <> FL PP ED

WN
Window number to which command applies (defaults to 1).

Lab
Label to define the desired type of zoom:

OFF
Turns zoom off (refits image of entire model to the window).

BACK
Goes back to previous zoom setting (five successive back ups, maximum).

SCRN
Interprets X1,Y1 as the screen coordinates of the center of a square zoom region; X2,Y2 as the screen coordinates of a point on one side of that square.

RECT
Interprets X1,Y1 and X2,Y2 as the screen coordinates of two opposite corners of a rectangular zoom region.

Notes

Zooms (centers and magnifies) the specified region of a display window. **/ZOOM** will operate on a display that has been formed by an explicit graphics action command (**APLOT**, **EPLOT**, etc.). **/ZOOM** has no effect on an "immediate" graphics display. When **/ZOOM** is executed, the display is automatically replotted such that the specified zoom region is centered and sized to fill the window.

Auto resizing is disabled when you issue the **/ZOOM** command. To restore auto resizing, issue the **/AUTO** command, or select **FIT** from the **Pan, Zoom, Rotate** box.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

Index

Symbols

A

B

C

D

E

F

G

H

I

J

K

L

M

N

O

P

Q

R

S

T

U

V

W

X