

ANSYS Troubleshooting Guide

ANSYS Release 8.1

001972
April 2004

ANSYS, Inc. is a
UL registered
ISO 9001: 2000
Company

ANSYS Troubleshooting Guide

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
001624	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

ANSYS, DesignSpace, DesignModeler, ANSYS DesignXplorer VT, ANSYS DesignXplorer, ANSYS Emax, ANSYS Workbench environment, CFX, AI*Environment, CADOE and any and all ANSYS, Inc. product names referenced on any media, manual or the like, are registered trademarks or trademarks of subsidiaries of ANSYS, Inc. located in the United States or other countries.

Copyright © 2004 SAS IP, Inc. All rights reserved. Unpublished rights reserved under the Copyright Laws of the United States.

ANSYS, Inc. is a UL registered ISO 9001: 2000 Company

ANSYS Inc. products may contain U.S. Patent No. 6,055,541

Microsoft, Windows, Windows 2000 and Windows XP are registered trademarks of Microsoft Corporation.

Inventor and Mechanical Desktop are registered trademarks of Autodesk, Inc.

SolidWorks is a registered trademark of SolidWorks Corporation.

Pro/ENGINEER is a registered trademark of Parametric Technology Corporation.

Unigraphics, Solid Edge and Parasolid are registered trademarks of Electronic Data Systems Corporation (EDS).

ACIS and ACIS Geometric Modeler are registered trademarks of Spatial Technology, Inc.

"FLEXIm License Manager" is a trademark of Macrovision Corporation.

Other product and company names mentioned herein are the trademarks or registered trademarks of their respective owners.

This ANSYS, Inc. software product and program documentation is ANSYS Confidential Information and are furnished by ANSYS, Inc. under an ANSYS software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, warranties, disclaimers and remedies, and other provisions. The Program and Documentation may be used or copied only in accordance with the terms of that license agreement.

See the ANSYS, Inc. online documentation or the ANSYS, Inc. documentation CD for the complete Legal Notice.

If this is a copy of a document published by and reproduced with the permission of ANSYS, Inc., it might not reflect the organization or physical appearance of the original. ANSYS, Inc. is not liable for any errors or omissions introduced by the copying process. Such errors are the responsibility of the party providing the copy.

Table of Contents

- 1. **Introduction** 1-1
- 2. **ANSYS Error Messages** 2-1

Chapter 1: Introduction

The Purpose of This Manual

The ANSYS Troubleshooting Guide is a compilation of common error messages and resolution recommendations. You can use this guide when you receive an error message that either stops your analysis, or does not provide a logical follow up to the events that preceded the message. The explanations and responses provided in this guide supplement the available documentation by providing information that is relative to the situation you are in when you see the error message.

Although only a portion of the ANSYS error messages are contained in this guide, using the online documentation's text search capability will often lead you to a message/resolution set that may provide some insight to the nature of your problem. The following section details how to access the resolutions via the ANSYS online documentation.

Accessing the Troubleshooting Information

You can access the troubleshooting information either directly from the help menu or by using the ANSYS Documentation search page.

The following section details the layout of the error messages that are contained in the Troubleshooting Guide.

Routine name and number

This is an internal code designator that uniquely identifies this error message. Although not accessible from within the software, this information can be helpful if you must contact ANSYS.

Error message string

This is the actual message that appears in the error message dialog box. This field will provide search criteria that you enter into the help system's search engine. You can use any portion of the string to generate a search.

Recommendations for resolution of the error message

Where possible, this information attempts to address the series of events or conditions that caused the error message.

Chapter 2: ANSYS Error Messages

Error Code: ChapHemi.1010

View Factor calculation stopped. Recompute View Factors for Radiation Problem.

You get this message when an analysis is stopped using the **STOP** button on the progress bar. The reason for stopping the analysis will determine the corrective action you will need to take. You can use the "Session Editor" to reissue the command that preceded the VIEW FACTOR CALCULATION. This could be **VFCALC** in **/AUX12**, or the **SOLVE** command in **SOLU**.

Closed_Cor.5010

PSD file has been lost.

You get this message when ANSYS is unable to find the database file **jobname.psd**. It is possible that the file was deleted, moved, or overwritten with another jobname file. If the PSD file cannot be recovered, the analysis must be restarted.

Closed_Ord.5010

PSD file has been lost.

You get this message when ANSYS is unable to find the database file **jobname.psd**. It is possible that the file was deleted, moved, or overwritten with another jobname file. If the PSD file cannot be recovered, the analysis must be restarted.

Closed_Ord.5050

No correlation between nodal and base excitations is allowed for table numbers %I:1 and %I:2.

This is a data entry mistake. For this case, it is physically impossible to define a correlation coefficient between the nodal and base excitations. Remove the correlation coefficient.

ConGrd.5010

Conjugate gradient solver did not meet specified convergence criterion within specified number of solver sweeps. Model size may require more sweeps. Check debug file for divergence. %/ %C:1.

In large problems, the default value of 500 sweeps is sometimes not enough. Increasing the number from 500 to 5000 will sometimes help. If this message occurs in the later stages of a solution, restart from a set of results just before the message was encountered, allowing more sweeps. In some cases, the model may be ill configured and unsolvable.

Contrl.5030

No change from initial solution - probable input error.

This message occurs when the initial solution satisfies all of the boundary conditions; no further iterations are required. This can happen when you forget to specify gravity or variable density for a natural convection problem. Verify the formulation of the analysis, make the proper additions and start the analysis over.

GetICV.5000

The following node(s) have their Initial mass fraction out of range. (Use the IC command).

The initial conditions you specified for species transport capability are in error. These values must be between 0 and 1 (inclusive). The sum of all applied species fractions at a node must also be between 0 and 1.

GetICV.5020

The following node(s) have their Initial mass fraction out of range. (Use the IC command).

The initial conditions you specified for species transport capability are in error. These values must be between 0 and 1 (inclusive). The sum of all applied species fractions at a node must also be between 0 and 1.

GmRes.5030

Coefficient matrix has a zero diagonal. Probably caused by conductivity or viscosity going to zero. Could also be caused by a divergent solution. %/ %C:1 %l:2.

If this message is encountered early in the solution, there is a problem with the way the analysis is set up. If it is encountered part way through the solution, the analysis needs to be solved better. Restarting from the set of results just before the divergence and specifying tighter convergence for each iteration (pressure or temperature sub iterations) may allow convergence.

GmRes.5040

Coefficient matrix has a negative diagonal. Probably indicates divergent solution. %/ %C:1 %l:2.

If this message is encountered early in the solution, there is a problem with the way the analysis is set up. If it is encountered part way through the solution, the analysis needs to be solved better. Restarting from the set of results just before the divergence and specifying tighter convergence for each iteration (pressure or temperature sub iterations) may allow convergence.

GmRes.5000

Generalized minimum residual solver did not meet specified convergence criterion within specified number of sweeps. Model size may require more sweeps. Check debug file for divergence. %/ %C:1.

In large problems, the default value of 500 sweeps is sometimes not enough. Increasing the number from 500 to 5000 will sometimes help. If this message occurs in the later stages of a solution, restart from a set of results just before the message was encountered, allowing more sweeps. In some cases, the model may be ill configured and unsolvable.

GmRes.5020

Generalized minimum residual solver did not meet specified convergence criterion within specified number of sweeps. Model size may require more sweeps. Check debug file for divergence. %/ %C:1.

In large problems, the default value of 500 sweeps is sometimes not enough. Increasing the number from 500 to 5000 will sometimes help. If this message occurs in the later stages of a solution, restart from a set of results just before the message was encountered, allowing more sweeps. In some cases, the model may be ill-configured and unsolvable.

InField.5030

A degree of freedom that was found in the .rfl file is not required for this analysis. Will not read .rfl values of %/ %C:1.

This is a data structure construction error message that means extra degrees of freedom (that are unused) were found in the results file, and that they will be ignored. Unused degrees of freedom can be created when you resume an old version of the data base before the species are added.

JacRatioWarn.3005

%C:1 element %I:2 has a zero or inverted Jacobian determinant at one of its corner nodes. Midside nodes, if any, may be poorly positioned.

This message indicates a geometry or element shape problem. An unacceptably shaped element is being stored (or one that is already stored is being tested). If the element was produced automatically (via the mesher or **VMESH**) generating a finer mesh may eliminate the problem, otherwise, the geometry may require modification, simplification or repair. Avoid curved-sided elements that span more than a 90 degree arc.

JacRatioWarn.3010

%C:1 element %I:2 has a Jacobian ratio of %G:3, which exceeds the warning limit of %G:4. Midside nodes, if any, may be poorly positioned.

This message indicates a geometry or element shape problem. An unacceptably shaped element is being stored (or one that is already stored is being tested). If the element was produced automatically (via the mesher or **VMESH**) generating a finer mesh may eliminate the problem, otherwise, the geometry may require modification, simplification or repair. Avoid curved-sided elements that span more the a 90 degree arc.

JacRatioWarn.3020

%C:1 element %I:2 has a zero or inverted Jacobian determinant at one of its corner node s. This may be OK if you are simulating a stress singularity using quarter point elements.

This message indicates a geometry or element shape problem. An unacceptably shaped element is being stored (or one that is already stored is being tested). Singular elements generated using **KSCON** will also initiate this message. If the element was produced automatically (via the mesher or **VMESH**) generating a finer mesh may eliminate the problem, otherwise, the geometry may require modification, simplification or repair. Avoid curved-sided elements that span more the a 90 degree arc.

JacRatioWarn.3030

%C:1 element %I:2 has a Jacobian ratio of %G:3, which exceeds the error limit of %G:4. This may be OK if you are simulating a stress singularity using quarter point elements.

This message indicates a geometry or element shape problem. An unacceptably shaped element is being stored (or one that is already stored is being tested). Singular elements generated using **KSCON** will also initiate this message. If the element was produced automatically (via the mesher or **VMESH**) generating a finer mesh may eliminate the problem, otherwise, the geometry may require modification, simplification or repair. Avoid curved-sided elements that span more the a 90 degree arc.

JacRatioWarn.3040

%C:1 element %I:2 has a Jacobian ratio of %G:3, which exceeds the error limit of %G:4. Midside nodes, if any, may be poorly positioned.

This message indicates a geometry or element shape problem. An unacceptably shaped element is being stored (or one that is already stored is being tested). If the element was produced automatically (via the mesher or **VMESH**) generating a finer mesh may eliminate the problem, otherwise, the geometry may require modification, simplification or repair. Avoid curved-sided elements that span more than a 90 degree arc.

Memory Error Messages

See ANSYS Memory Error Messages in the *ANSYS Basic Analysis Guide* for a complete list of memory error messages.

MidParOffsWarn.3040

%C:1 element %I:2 has a midside node which is off-center by a distance ratio of %G:3. This may be OK if you are simulating a stress singularity using quarter point elements.

This message indicates a geometry or element shape problem. An unacceptably shaped element is being stored (or one that is already stored is being tested). If the element was produced automatically (via the mesher or **VMESH**) generating a finer mesh may eliminate the problem, otherwise, the geometry may require modification, simplification or repair. Avoid curved-sided elements that span more than a 90 degree arc.

NNewVisLaw.5020

Non-Newtonian viscosity computed to be non-positive.

This message probably means that the coefficients specified for the non-Newtonian viscosity variations are wrong, since the value has come out to be zero or less, which is meaningless. If it happens right away, it indicates an input error. If it happens further on into the solution, it probably indicates divergence.

NNewVisLaw.5040

Non-Newtonian viscosity computed to be unreasonable, please check coefficients.

This message probably means that the coefficients specified for the non-Newtonian viscosity variations are wrong, since the value has come out to be zero or less, which is meaningless. If it happens right away, it indicates an input error. If it happens further on into the solution, it probably indicates divergence.

PropCond.5000

Negative or zero temperature detected. This may be caused by a diverging solution or by an incorrect temperature or property value. A temperature offset may also be required. (TOFFST) %I %C:1 %I:2.

Your temperature has dipped below zero. If this message is encountered early on in the analysis, it is probably due to some sort of bad input (lack of a **TOFFST**, too low a reference temperature). Later on in the analysis, it could indicate divergence. There are a number of ways to avoid this condition: 1) turn on capping for temperature or pressure until the values stabilize, 2) use a transient algorithm, 3) increase your reference temperature/pressure, 4) refine your mesh around the problem area.

PropCond.5010

Negative conductivity calculated. Solution terminated. This could be caused by a property error or divergent solution. %/ %C:1 %I:2.

If your conductivity is temperature dependent and the temperature dips below zero, an invalid conductivity value will be generated. If this message is encountered early on in the analysis, it is probably due to some sort of bad input (lack of a **TOFFST**, too low a reference temperature). Later on in the analysis, it could indicate divergence. There are a number of ways to avoid this condition: 1) turn on capping for temperature or pressure until the values stabilize, 2) use a transient algorithm, 3) increase your reference temperature/pressure, 4) refine your mesh around the problem area.

PropDens.5030

Negative density calculated. Solution terminated. This could be caused by a property error or a divergent solution. %/ %C:1 %I:2.

When this message occurs early in the solution, it probably means that the properties were specified incorrectly. When it occurs after a few iterations, it probably means divergence (which could also be due to a bad specification). You can switch variable density **OFF** to allow any swings in the solution to stabilize, and then restart the analysis.

PropSpHt.5000

Negative or zero temperature detected. This may be caused by a diverging solution or by an incorrect temperature or property value. A temperature offset may also be required. (TOFFST) %/ %C:1 %I:2.

Your temperature has dipped below zero. If this message is encountered early on in the analysis, it is probably due to some sort of bad input (lack of a **TOFFST**, too low a reference temperature). Later on in the analysis, it could indicate divergence. There are a number of ways to avoid this condition: 1) turn on capping for temperature or pressure until the values stabilize, 2) use a transient algorithm, 3) increase your reference temperature/pressure, 4) refine your mesh around the problem area.

PropSpHt.5010

Negative or zero pressure detected during property update. This may be caused by a divergent solution or by an incorrect property value. The reference pressure may need to be increased. %/ %C:1 %I:2.

Your pressure has dipped below zero. This is problematic since later on in the analysis, density will be updated as a negative number for compressible flows. If this message is encountered early on in the analysis, it is probably due to some sort of bad input (lack of a **TOFFST**, too low a reference temperature). Later on in the analysis, it could indicate divergence. There are a number of ways to avoid this condition: 1) turn on capping for temperature or pressure until the values stabilize, 2) use a transient algorithm, 3) increase your reference temperature/pressure, 4) refine your mesh around the problem area.

PropSpHt.5020

Negative Specific Heat calculated. Solution terminated. This could be caused by a property error or divergent solution. %/ %C:1 %l:2.

This message can occur when you have temperature dependent specific heats -- a negative specific heat can occur when the temperature dips below zero or goes out of range of the specified curve /table of specific heat vs. temp. If this message is encountered early on in the analysis, it is probably due to some sort of bad input (lack of a **TOFFST**, too low a reference temperature). Later on in the analysis, it could indicate divergence. There are a number of ways to avoid this condition: 1) turn on capping for temperature or pressure until the values stabilize, 2) use a transient algorithm, 3) increase your reference temperature/pressure, 4) refine your mesh around the problem area.

PropVisc.5000

Negative or zero temperature detected. This may be caused by a diverging solution or by an incorrect temperature or property value. A temperature offset may also be required. (TOFFST).

If this message occurs early in the solution, it probably indicates a bad input, such as forgetting to specify **TOFFST**, or, in a compressible case, a mistake in the input calculations that sets up an impossible flow. If it occurs later on in the solution, it probably indicates divergence.

PropVisc.5010

Negative viscosity calculated. Solution terminated. This could be caused by a property error or divergent solution. %/ %C:1 %l:2.

If this message occurs early on in the solution, it probably indicates an input problem. If it occurs later on in the solution, the analysis is probably divergent.

PrpCals.5010

Negative or zero molecular weight found for species calculations. Solution terminated. %/ %C:1 %l:2.

This message is due to an input error, since the molecular weight is an input quantity. Please check your input data and try the analysis again.

accat3.5000

Data mismatch during ACCAT operation.

Some previous operation stored inconsistent area or line data. Identifying where or how this occurred is difficult. Often, this condition can be corrected by a **SAVE/RESUME** cycle, or by saving, exiting ANSYS and restarting. You can also perform a **CDWRITE/CDREAD** cycle. Using the **CDWRITE/CDREAD** or **SAVE/RESUME** operations is often more effective and less problematic than deleting and redefining the incorrect entities.

ardel2.5020

A data mismatch occurred during the ADELET operation.

Some previous operation stored inconsistent area or line data. Identifying where or how this occurred is difficult. Often, this condition can be corrected by a **SAVE/RESUME** cycle, or by saving, exiting ANSYS and restarting. You can also perform a **CDWRITE/CDREAD** cycle. Using the **CDWRITE/CDREAD** or **SAVE/RESUME** operations is often more effective and less problematic than deleting and redefining the incorrect entities.

bacsub.3000**The sparse solvers are not available with 5.2 CE logic. %/ The SOLUTION is abandoned.**

This message is produced when **/CONFIG,CEONFULL,1** is active and the solution is attempted with the sparse solver (**EQSLV,SPARSE**). The configuration you have chosen for your analysis is not suitable for this solver, the solution should be performed with the Frontal Solver.

bacsub.3010**The multiphysics JCG/ICCG solvers are not available with 5.2 CE logic. %/ The SOLUTION is abandoned.**

This message is produced when **/CONFIG,CEONFULL,1** is active and the solution is attempted with the either the JCG, or ICCG solver. The configuration you have chosen for your analysis is not suitable for this solver, the solution should be performed with the Frontal Solver.

bcssl4.3000**There are no active degrees of freedom.**

This message indicates that all Degrees of Freedom in the model are constrained, or that no elements have been selected in the current set when the **SOLVE** command is issued. Although total DOF constraint can be acceptable, you should check the application of forces and constraints and the selection of elements for the solution.

bcssl4.3070**Memory used at the current substep exceeds that of the previous step in the Sparse Matrix Direct Solver linear loop. New factorization done.**

This is a Sparse Direct Solver message indicating that the number of nodes or elements has changed between iterative steps in the solution. The message alerts you to this and adjusts the matrix as needed.

bcsswn.3030**Error in numeric factorization. %/ The system matrix, possibly, is numerically singular.**

When you are in the Sparse Direct Solver, this message indicates that the model is not constrained. Verify your specifications for the **D** command.

bhconv.5000**Cannot convert B-H data.**

Check the data that you entered for the B-H curve. A B-H curve is represented by nonzero pairs of B and H values. If any one of these values is zero or negative, this error message will be displayed. Check your pairs of B and H data.

brkmsh.5000**Volume %l:1 could not be meshed because of a data storage fault (code = %l:2).**

This message can occur for a number of reasons: 1) If you attempt to mesh a hex-shaped volume with bricks and assign element division that do not match any of the prescribed transition patterns (Modeling and Meshing Guide, Chapter 7). 2) The process ran out of memory. 3) **VMESH** or some earlier operation corrupted the data for this volume. 4) The user aborted meshing with the **STOP** button. You may need to go back and redefine the mesh according to documented methods, or redo the **VMESH** operation. Mesh generation problems can also be rectified by performing a **SAVE/RESUME** or **CDWRITE/CDREAD** cycle. Using **SAVE/RESUME** frequently during the analysis will help to prevent or rectify incorrect database write operations.

btcfm.5000**Meshing failure in volume %l:1 because mesher reports a maximum output node number of %l:2, but space is only allocated for %l:3 nodes.**

You get this message when ANSYS incorrectly allocates memory for a brick mesh. Try a different combination of element divisions or a different transition pattern. Using **SAVE/RESUME** will sometimes help to eliminate incorrect entities, and is more effective than deleting and redefining incorrect entities.

btcfm.5010**Meshing failure in volume %l:1 because mesher reports a maximum output element number of %l:2, but space is only allocated for %l:3 elements.**

You get this message when ANSYS incorrectly allocates memory for a brick mesh. Try a different combination of element divisions or a different transition pattern. Using **SAVE/RESUME** will sometimes help to eliminate incorrect entities, and is more effective than deleting and redefining incorrect entities.

btcmstr.5015**Brick meshing failure in volume %l:1 because the assigned element divisions do not match a recognized transition pattern.**

You need to check the way you have assigned the transition patterns when you try to brick mesh a hex-shaped volume. Chapter 7 of the Modeling and Meshing Guide provides recommended patterns.

btcmstr.5020**Meshing failure in volume %l:1 because of confusion about the number of areas on the volume.**

VMESH or some earlier operation may have corrupted the data for this volume. A **SAVE/RESUME** cycle can often rectify this problem; sometimes it may be necessary to **SAVE**, restart ANSYS and then **RESUME**.

cad_area.5000**Area External Loop Specification Error. %/ Area can not be stored.**

There is a problem with an area that was transferred or imported from an external CAD package or connection product. An error like this cannot be resolved within ANSYS. Go back to your original model and inspect the area of concern for redundancies, open loops or other import degeneracies.

cad_fil_srf.5000**Fillet contains a degenerate point. %/ Area can not be stored.**

There is a fillet area or surface representation being imported that is not supported by the external CAD product. Try going back to your CAD package and simplifying or modifying the fillet area.

cad_volu.5000**At least 1 area was unable to be stored. %/ Volume can not be stored.**

An area in your model was unable to be stored in the ANSYS database. This is probably due to a twisted surface or an indeterminate projection result. The lines that bound an area are sometimes projected onto an uneven surface (twisted) and can create geometric anomalies. You can try going back into **/PREP7** and issuing **LSEL, ,EXT**. This will select only the lines that do not share 2 areas. You can then try to form acceptable areas from those lines by closing the open areas.

cad_volu.5010**At least 1 area was unable to be stored. %/ Volume can not be stored.**

An area in your model was unable to be stored in the ANSYS database. This is probably due to a twisted surface or an indeterminate projection result. The lines that bound an area are sometimes projected onto an uneven surface (twisted) and can create geometric anomalies. You can try going back into **/PREP7** and issuing **LSEL, ,EXT**. This will select only the lines that do not share 2 areas. You can then try to form acceptable areas from those lines by closing the open areas.

ckmgev.5000**Unable to evaluate nonlinear magnetic curve. Element= %I:1 Material= %I:2 Abscissa= %G:3.**

The B-H data you entered could not be converted to properly fit a spline curve. Without the spline curve, evaluation of intermediate values of B-H data is not possible. Check the original B-H data and ensure that the data values are steadily increasing, and that the data plots are fairly smooth when displayed in B-H form.

d3msar.5000**Cannot plot area %I:1. %/ Surface tessellation error.**

Surface tessellation has failed. Try plotting the wireframe of the geometries by using **/FACET,wire**. You should also examine the problematic area and make the necessary modifications to the area (simplify the geometry, verify connectivity, etc.)

d3msar.5010**Error in surface tessellation for area %I:1. This area can only be displayed with wire frame, and cannot be part of a V, VA, VSUM or ASUM operation.**

Surface tessellation has failed. Try plotting the wireframe of the geometries by using **/FACET,wire**. You should also examine the problematic area and make the necessary modifications to the area (simplify the geometry, verify connectivity, etc.)

eoelem.5010

Please report this system error from eoelem to your ASD or ANSYS, Inc. Try another element option. Error harmful only on parallel machines. RST element record size from FILLEN does not match actual; %/ FILLEN= %! :1 ACTUAL= %! :2 Element Type= %! :3 Element= %! :4.

This is parallel processor (NPROC greater than 1) problem. The element related files esav, emat and rst are written to in a random (not serial) order. The write destination for the element information must be predefined before you begin element processing. If the information is written incorrectly, another element's data could be overwritten. Try to run the combination of options with only one processor active (NPROC = 1).

ernman.5000

Topological degeneracy detected for %C:1 command. Try modifying geometry slightly or relaxing the tolerance (BTOL command). If BTOL is relaxed, be sure to change the tolerance back to the default after operation.

You will get this message during Boolean operations. You can modify the geometry or relax the tolerance, as noted. Boolean operations are often dependent on the order in which they are performed. Check your model and perhaps modify the order of operations in order to obtain a satisfactory result.

errasd.5000

Unable to complete Boolean operations. Try modifying geometry slightly or adjusting the tolerance value on BTOL command. If changed, return BTOL to default value after completing operation.

You will get this message during Boolean operations. You can modify the geometry or relax the tolerance, as noted. Boolean operations are often dependent on the order in which they are performed. Check your model and perhaps modify the order of operations in order to obtain a satisfactory result.

errdeg.5000

Parametric degeneracy detected for %C:1 command. Try modifying geometry slightly or relaxing the tolerance (BTOL command). If BTOL is relaxed, be sure to change the tolerance back to the default after operation.

You will get this message during Boolean operations. You can modify the geometry or relax the tolerance, as noted. Boolean operations are often dependent on the order in which they are performed. Check your model and perhaps modify the order of operations in order to obtain a satisfactory result.

lanbwn.5000

Eigenvalue extraction failed, possibly because mass matrix is non-definite (has both positive and negative definite parts).

MODOPT,LANB uses the Block Lanczos Eigensolver. In a modal analysis, this message indicates a zero or negative diagonal. You can use the **PIVCHECK** command to override the message and continue the analysis.

lanbwn.5010

Eigenvalue extraction failed. Singular matrix encountered repeatedly in the iteration loop, possibly because it is an ill-posed eigenproblem.

This message is due to a poorly configured modal analysis. The model has no constraints, which causes zero eigenvalues to be part of the eigenvalue solution for the rigid body modes. Although this is a normal situation, when combined with a mass matrix zero diagonal, the solution values can be incorrect. You can try to add the appropriate constraints to the model and then restart you analysis.

lantri.5030

There are %I:1 positive pivots with ratios less than %G:2 indicating a possible poorly conditioned stiffness matrix.

This message indicates that portions of your model are extremely soft, while others are extremely stiff. Check the Young's Modulus (**MP,EX**) specified for the structural members to ensure valid material properties.

lantri.5040

There are %I:1 negative pivots with ratios greater than %G:2 indicating a possible poorly conditioned stiffness matrix.

This message indicates that portions of your model are extremely soft, while others are extremely stiff. Check the Young's Modulus (**MP,EX**) specified for the structural members to ensure valid material properties.

lccat3.5000

A data mismatch occurred during the LCCAT operation.

Some previous operations stored inconsistent area or line data, making it impossible to complete the requested **LCCAT** operation. Often, this problem can be rectified by a **SAVE/RESUME**, or **CDWRITE/CDREAD** cycle.

magpr.5010

For material %I:1 the nurb representation of the B-H curve could not be constructed properly.

The B-H data you entered could not be converted to properly fit a spline curve. Without the spline curve, evaluation of intermediate values of B-H data is not possible. Check the original B-H data and ensure that the data values are steadily increasing, and that the data plots are fairly smooth when displayed in B-H form.

mainst.200

ANSYS could not initialize the high level memory manager. Error code= %I:1.

This message is returned when a malloc (memory allocation) call is returned NULL. You can try closing all other applications and restarting ANSYS, or you can change your -m specification. It is possible that the -m specification is so large that there is not enough left to start the memory manager. Verify the amount of memory available to your system and check your run statistics to verify the viability of your analysis.

mainst.400**ANSYS could not initialize the heap memory manager. Error code= %l:1.**

This message is returned when a malloc (memory allocation) call is returned NULL. You can try closing all other applications and restarting ANSYS, or you can change your -m specification. It is possible that the -m specification is so large that there is not enough left to start the memory manager. Verify the amount of memory available to your system and check your run statistics to verify the viability of your analysis.

mlccat3.5000**A data mismatch occurred during the LCCAT operation.**

You get this message when inconsistent area or line data has been stored by a previous operation. Using **SAVE/RESUME** (and sometimes restarting in between) can often fix these kinds of improper write operations.

p6tfun.5010**Transfer function can not be computed for the %/ Wave propagation or spatially correlated case.**

You cannot request a transfer function for an excitation analysis involving wave propagation or spatial correlation. If a transfer function is desired, the analysis should be run as an ordinary excitation analysis.

p6tfun_close.5010**Transfer function can not be computed for the %/ Wave propagation or spatially correlated case.**

You cannot request a transfer function for an excitation analysis involving wave propagation or spatial correlation. If a transfer function is desired, the analysis should be run as an ordinary excitation analysis.

pOutputLoop.5000**Please report this system error from pOutputLoop to your ASD or ANSYS, Inc. Try another element option. Error harmful only on parallel machines. RST element record from FILLEN does not match actual %/ FILLEN= %l:1 ACTUAL= %l:2 Element Type= %l:3.**

This is parallel processor (NPROC greater than 1) problem. The element related files esav, emat and rst are written to in a random (not serial) order. The write destination for the element information must be predefined before you begin element processing. If the information is written incorrectly, another element's data could be overwritten. Try to run the combination of options with only one processor active (NPROC = 1).

psdcom.5020**Missing element order vector in psdcom. Please send your data to your ANSYS Technical Support Provider.**

You get this message when the element order vector cannot be located in the ANSYS database. This could occur when there is not enough virtual memory available to generate the vector. You should attempt to increase the virtual memory setting for the machine you are using and restart the analysis. It is also a good idea to check the run statistics before initiating the analysis. If the problem persists, send the model and data files to your ASD.

psdfit.9200

The design set data resulted in a singular Least Square problem. %/ Please check your data.

You get this message when two points in the input PSD chart are too close together. Each segment of the PSD chart is transformed into a polynomial through the least square method. During this operation, some intermediate points in a segment are selected to constitute a design matrix for solving the polynomial coefficients. If the segment is very small and the boundary points are too close, then a singularity may occur due to machine precision. This condition can be rectified by redefining the PSD chart and ensuring that the points are not too close together.

qtcstr.5000

An error has occurred in routine ESZSET. Please provide this information to your ANSYS Support Distributor.

You get this message when inconsistent area or line data has been stored by a previous operation. Using **SAVE/RESUME** (and sometimes restarting in between) can often fix these kinds of improper write operations.

qtrmid.5000

An error has occurred in generating the midside nodes. Please provide this information to your ANSYS Support Distributor.

You get this message when inconsistent data is generated by one of the area or volume mesh routines. Using **SAVE/RESUME** (and sometimes restarting in between) can often fix these kinds of improper write operations.

qtrmsh.5070

Area %l:1 could not be meshed because of a data storage fault (code = %l:2).

You get this message when inconsistent area or line data has been stored by a previous operation. Using **SAVE/RESUME** (and sometimes restarting in between) can often fix these kinds of improper write operations.

sasdof.3000

Fail to transfer symmetry or anti-symmetry constraints to node %l:1. %/Contact the technical support.

A line or area symmetry or anti-symmetry constraint is being transferred to the nodes, making it impossible to resolve the constraint directions as specified. Try deleting one or more of the line or area constraints, and then substituting direct nodal constraints in the specified direction. If all of the specified symmetry conditions make sense, then report this error to your ASD.

sfelem.5010

Please report this system error from sfelem to your ASD or ANSYS, Inc. Try another element option. Error harmful only on parallel machines. ESAV record size error by FILLEN; %/ FILLEN= %l:1, ACTUAL= %l:2, Element Type= %l:3.

This is parallel processor (NPROC = greater than 1) problem. The element related files esav, emat and rst are written to in a random (not serial) order. The write destination for the element information must be predefined before you begin element processing. If the information is written incorrectly, another element's data could be overwritten. Try to run the combination of options with only one processor active (NPROC = 1).

sfelem.5020**EMAT record size error in FILLEN %/ FILLEN = %l:1 ACTUAL= %l:2 JTYPE= %l:3.**

This is parallel processor (NPROC = greater than 1) problem. The element related files **esav**, **emat** and **rst** are written to in a random (not serial) order. The write destination for the element information must be predefined before you begin element processing. If the information is written incorrectly, another element's data could be overwritten. Try to run the combination of options with only one processor active (NPROC = 1).

slvdfs.5095**IT IS HIGHLY RECOMMENDED TO USE NEW SOLU CONTROL OPTION BY ISSUING SOLCON,ON, FOR THIS PROBLEM FOR ROBUSTNESS AND EFFICIENCY.**

You will get this message if you set **SOLCONTROL,OFF**. We strongly recommend that you use **SOLCONTROL,ON** for this analysis.

sysopn.5000**Unit number [%l:1] exceeds maximum unit number [%l:2]**

This is a "User Programmable Feature" message indicating that a specified Fortran unit number is greater than the value allowed by ANSYS. The maximum unit number is 60.

tbtabs.5020**Table is too large to edit.**

The maximum table size that can be handled by the array editor is 250 rows and/or 100 columns. Edit the table directly using the **TBMODIF**, **TBPT**, or **TBDATA** commands.

veg_bkte_compare.5060**There is a problem with the adjacent element data for element %l:1 at face %l:2 (and possibly elsewhere). Validation of mesh connectivity is incomplete.**

You get this message when the adjacent element logic for the **MCHECK** operation is invalid. You can sometimes rectify this problem by performing a **SAVE/RESUME** cycle, or by using **CDWRITE/CDREAD** (you may need to restart ANSYS in between).

veg_bkte_compare.5070**The volume of element %l:1 (and possibly others) is undefined. Validation of mesh connectivity is incomplete.**

You get this message when the **MCHECK** logic returns an invalid result, or cannot be completed. Try performing a **CHECK** operation before **MCHECK**.

veg_qtri_compare.5040

There is a problem with the adjacent element data for element %l:1 at face %xl (and possibly elsewhere). Validation of mesh connectivity is incomplete.

You get this message when the adjacent element logic for the **MCHECK** operation is invalid. You can sometimes rectify this problem by performing a **SAVE/RESUME** cycle, or by using **CDWRITE/CDREAD** (you may need to restart ANSYS in between).

veg_qtri_setup.5010

The normal ment %l:1 (and possibly others) is undefined. Validation of mesh connectivity is incomplete.

The **MCHECK** operation could not be completed. Check the connectivity of your mesh.

vlfl28.5000

Data mismatch reading VA volume %l:1.

This message is generated when the data input with the **VA** command is incorrect. Check the input data and retry the command.