Troubleshooting and FAQ 10

This chapter contains two sections, frequently asked questions (FAQ) and a troubleshooting section that describes some common problems that may be encountered while installing or using MSC/NASTRAN for Windows (MSC/N4W), with avoidances and fixes. This troubleshooting section is broken up into sections, based upon when problems may arise:

- during installation
- starting up the software
- running the modeler program
- running an analysis

Problems that occur on one Windows system but not another are noted whenever possible. In many cases, the symptom of the problem is a Dialog Box, or pop-up Window, which gives a short description of the problem. These are indicated by the word *Dialog:* in the sections below.

10.1 Troubleshooting

10.1.1 Problems During Installation/Configuration

The following is a list of the problems that can occur during installation and configuration of MSC/N4W. In general, the installation and configuration process should go smoothly, the process is described in complete detail in Chapter 2. However, some situations where problems could arise have been identified below.

10.1.1.1 Dialog: Setup needs NNN KB of disk space

Problem:

The installation Setup program needs a small amount of temporary disk space while it is running. The program will use the directory specified by Windows TEMP variable (usually c:\temp or c:\windows\temp). This will be necessary even if you are installing on a drive other than the c: drive.

Solution:

Make additional space available on the drive where the TEMP directory is location, or change the TEMP directory definition. Changing the TEMP variable can usually be done in the **System** section of the **Control Panel**.

10.1.2 Problems Starting Software

There are many problems that could occur while starting up the software. These will generally be accompanied by a dialog box that briefly describes the problem.

10.1.2.1 Program does not start - no messages

Problem:

If you have any difficulty starting MSC/N4W, either the program will not start, or an abnormal termination occurs, you either do not have enough disk space to create the temporary files that MSC/N4W needs while it is running, or your Windows TEMP directory is not set to a valid, accessible directory.

Solution:

You may either change your Windows TEMP directory, or specify a path for the MSC/N4W scratch files (which default to the Windows TEMP directory) to a valid directory. The scratch file location is changed in **File | Preferences**.

10.1.2.2 Dialog: Cannot start mplayer - Error 2

Problem:

The software does not start up when you select the program from the **Start** button (or **Program Manager**). Instead a dialog comes up with the message above.

Solution:

Find out where the file `mn4w3dev.ini` is located on your system. It should be put in the "Windows" directory. This directory is usually `c:\windows`, but could be another directory, depending on your system.

This file should contain the information described in Chapter 2. If this file does not exist, or does not have the correct information, the best action to take is to reinstall the software.

10.1.3 Problems Running Modeler

10.1.3.1 Poor graphics quality in render mode

Problem:

Displaying the model in Render mode gives highly dithered, poor quality images.

Solution:

The color mode for your system is not in a high-color mode. Change the color mode from 16-color or 256-color to 32,000 color, 64,000 color, True Color, etc. This can be done in **Control Panel | Display**.

10.1.3.2 Graphics “flashing” during dynamic rotation

Problem:

Flashing occurs during model dynamic rotation, panning, etc.

Solution:

Change the setting in **File | Preferences | Views | Dynamic** to a different variable. One of the settings will work best for your graphics driver.

10.1.3.3 Opening models, importing and reading results is slow

Problem:

The process of opening a model, importing geometry or analysis models, and/or reading results after an analysis is slow.

Solution:

Change the **Cache Pages** setting in **File | Preferences | Database**.

10.1.4 Problems Running Analysis

When you run an analysis in MSC/N4W by selecting **File | Analyze**, a number of events take place. After entering all the information in the analyze menu and selecting the **OK** button, the following should happen:

1. The program should ask you to save the model you are working on.
2. The modeler window should be reduced to an icon or an entry on the **Start** menu.
3. The Manager window should be brought to the front, along with a black “DOS box” window.
4. The Manager window displays certain data for the duration of the analysis, until the message “Analysis is Complete” appears. The clock will update every few seconds.
5. The **Manager** window will reduce and the **Modeler** window will be brought to the maximum size, with the current model displayed.

If there is any breakdown in this chain of events, the analysis will not complete successfully. Situations where problems could arise have been identified below.

10.1.4.1 Manager windows starts but no messages are shown

Problem:

The **Manager** window starts, but does not seem to have any activity. Only the Clock may appear in the window, and this may seem to be frozen at one time. The **Modeler** window never restarts.

Solution:

Increase the amount of Virtual memory on your system. The guideline is at least 80 MB of virtual memory, but 100-120 MB is suggested. See Chapter 2 for more information on configuring Virtual Memory.

10.1.4.2 Dialog: Your Analysis Failed (Return Code = -9002)

Problem:

The **Manager** windows starts, flashes messages for a few seconds, then control returns to the Modeler, where the above message appears.

Solution:

This will occur any time there are spaces in either the model file name or the folder names in which the model file is contained. The same message may occur if you install the software in a folder with spaces (i.e., c:\Program Files). Make sure that the folders containing the model files and the software do not contain any spaces.

10.1.4.3 Dialog: Your Analysis Failed (Return code=-20)

Problem:

The **Manager** windows starts, scrolls messages for a period of time, and can often even display the "Analysis is Complete" message. However, when control returns to the Modeler, the above message appears.

Solution:

This will occur any time the analysis starts successfully (i.e. no authorization problems), but fails during the analysis execution. There could be a number of possible causes, such as modeling problems (incomplete materials or loads definition) or system problems (running out of disk space). Please enter **Yes** to this dialog box to continue to the solver **Message Review** for more information. Some common solver messages are included below.

10.1.5 Common Solver Messages

Upon completion of the analysis, after the modeler has restarted, the **Message Review** window will be displayed. The messages should *always* be reviewed, even if a successful analysis has been indicated. There is a **Help** button in this menu which displays further information on all of the solver messages.

The more common error message numbers that may be reported, with an explanation of the possible causes, are given below.

USER FATAL MESSAGE 9050 (SEKRRS) - RUN TERMINATED DUE TO EXCESSIVE PIVOT RATIOS

This error occurs when the model has not been restrained properly. In linear static analysis, sufficient restraints should be applied to the model to prevent rigid body motion. Check the applied restraints and correct them.

USER FATAL MESSAGE 4276 Error Code xxx.

This error indicates insufficient resources during the simulation process, usually disk space. Check to see if your disk drive has sufficient free space available for the analysis.

USER FATAL MESSAGE 5271.

The rest of the message is: The ratio of the longest edge to the shortest altitude exceeds 100.

This error shows that the quality of one or more elements in the simulation model is unacceptable to the MSC/NASTRAN solver. Try re-meshing the model with a different average element size.

USER FATAL MESSAGE 316 - Illegal data on Bulk Data Card XXXX

This will occur whenever you have “bad” data in your model definition. One common example is the incorrect definition of material or property data, such as defining a Young’s Modulus of 0.0.

The XXXX after Bulk Data Card will be replaced by a keyword. Refer to the *MSC/NASTRAN Encylopedia* for detailed information on MSC/NASTRAN Bulk Data.

USER WARNING MESSAGE 2148 - SPCD on a Point not in the S-set

This will occur when displacement loads (enforced displacements) have been defined that are not accompanied by a nodal constraint. The nodes which have displacement loads must have a constraint on the same Degrees of Freedom as the displacement load, otherwise the displacement loads are skipped, and the above message is issued.

USER WARNING MESSAGE 4420 - The following degrees of freedom are potentially singular

If this message is issued, it is unlikely that results will have been generated by the analysis, even though there are no FATAL messages issued. This message could occur because your model is under constrained, or you may have rigid body motion in your model.

10.2 Frequently Asked Questions

Question: I cannot see geometry that I have just created.

Two of the most common causes of this are using Render mode, and limiting your view to a group which is not the active group. Check for the following situations:

1. See if Render mode has been turned on, which can be enabled from the View Style button on the Toolbar or the Render checkbox on the View/Select menu. This mode enables dynamic hardware graphics display for OpenGL-compatible graphics cards, increasing the speed. However, if left on, this mode also requires the hardware's graphics buffer to be explicitly updated by entering Ctrl-G after new entities are added. Turning Render mode off, or explicitly entering the Ctrl-G when needed, will solve the problem.

2. See if your view of your model is set to a specific group that is not the current active group. When there is an active group with the Automatic Add option enabled, this problem can occur when another group being viewed. To check this, enter the View/Select option, pick the Model button, and temporarily set the Group setting to Active or None. If your new geometry then appears after closing the menu, you should change the group setting to Active or None on this menu.

Question: I am getting an "Excessive pivot ratio" error when I perform a structural analysis.

This problem is usually caused by having insufficient boundary conditions on a structural model. If a model is underconstrained, it can be free to move in space, a condition that will cause the above-mentioned “Excessive pivot ratio” error. Review your boundary conditions in MSC/NASTRAN for Windows using the List/Model/Constraint and/or Modify/Edit/Constraint menu options, and then go back and constrain the model as needed.

You may also have created the proper constraints, but they may reside in a constraint set you did not select or make active. The active constraint set is shown in a box in the lower right corner of the viewport. The box shows “con:#”. The # is the current constraint set, and clicking on this box will open a window that will list the names of all of the constraint sets in the model. The current constraint set will be displayed on the screen when you do a View/Regenerate, and it will use the constraint set that is automatically chosen when you do File/Analyze.

Another cause of Excessive Pivot Ratio is unwanted free edges in the model. This typically occurs at the boundaries between geometric entities. To remove the unwanted free edges one must first make sure that the meshes are congruent and then merge the coincident nodes. The models free edges may be viewed by selecting View/Select/Model Style/Free Edge.

Question: How can I move my MSC/NASTRAN for Windows software from one computer to another?

As of Version 4, a single set of authorization codes will permanently upgrade your security device (or “dongle”) for using both the modeling and solver portions of MSC/NASTRAN for Windows. Therefore, if you are currently running this product successfully on one machine, simple move the security device to the printer port of your new machine, and then install the software from your original CD-ROM on this machine.

Question: How can I change the color scheme of my contour spectrum? Can I set it to a grey scale?

Using the View/Options menu, select a Category field of PostProcessing, and then select Contour/Criteria Levels from the list on the left side of the menu box. Next, on the right side of this menu, set Contour Palette to User Palette. Then pick the Set Levels button to produce a submenu listing contour level values and colors. From there, you can use one of two approaches to set a new contour spectrum.

1. Pick the User Palette button, and from that menu, select the Load button to pick from a library of predefined contour palette files. A file browser will appear, and you will need to locate the Modeler subdirectory (such as C:\MSCN4W\Modeler) to list these palette files, which have a file extension of .cnt. Those files with “mono” in the filename contain monochromatic grayscale palettes.

2. For each contour level, selecting the Palette button next to it will allow you to graphically select a color value for this level, including both color and grey scale choices. You may also custom blend a color from component red, green and blue color values.

When finished, pick OK on both the Set Levels form and the View/Options form to set this new spectrum.

Question: How can I permanently turn off the rulers and workplanes for my model?

Workplanes and rulers can be toggled on and off via the View/Options menu, under the Tools and View Style category, by selecting Workplane and Rulers and then clicking off the Draw Entity button and saving the option. However, this does not permanently turn the option off for new models. To accomplish this, go to the File/Preferences menu, click on the Views button, and click on the button for Workplane Never Visible in New View. Save these preferences by picking OK on both menus, and when files are opened in the future, the rules and workplane will not appear.

Question: How can I change the position and scale of my workplane?

You can do this by either choosing the Tools/Workplane menu, or right clicking anywhere within your graphics window, and then selecting the Workplane option. Either selection will produce a menu with options to define scale, offset, rotate or align the workplane.

Question: Can I output my analysis results to a spreadsheet?

Yes. The File/Export/Analysis Model contains an option for exporting results in comma-separated values format, which will produce a .CSV file which can be read directly into spreadsheets such as Microsoft Excel.

Alternatively, you can also create a text file of results data for use in other applications, using the List/Output/XY Plot menu option. Simply select and plot the XY data you wish to capture using the standard View/Select options, set an output file name using the List/Destination option, and then perform the list output.

Question: How can I change my 4-noded tetrahedral elements to 10-noded tetrahedra?

You can do this using the Modify/Update Elements/Order menu, which will change the element order from linear to parabolic. Select the appropriate tetrahedra, pick OK on the selection dialog, and then select Yes to the subsequent prompt about converting the elements to parabolic. Delete all will not delete node being used by existing element.

You may also convert quadratic elements to linear using the same menu, in this case by answering No to the question about converting to parabolic form. Be aware, however, that this operation does not automatically delete the existing midside nodes. These nodes must be deleted separately. In cases like these, the Select All option on the Delete/Model/Node selection menu can be a convenient way of deleting all nodes not assigned to elements.

Question: How can I switch between different materials for certain elements after an analysis?

Use the Modify/Update Elements/Material ID menu to select elements, and then specify a new material ID from existing material definitions. This Modify/Update Elements menu also allow you to change many other element attributes such as element types, property IDs, material angles and beam offsets.

Question: Is it possible to merge two MSC/NASTRAN for Windows databases together?

Two or more MSC/NASTRAN for Windows databases may be merged by first exporting them as individual FEMAP Neutral Files, using the File/Export/FEMAP Neutral menu option, and then importing these files into a single database using the File/Import/FEMAP Neutral menu.

When merging these files into a single database, be sure that the ID values of individual entities from each database do not conflict - this can lead to database errors. The file import dialog screen contains a Model Merge Options/ID Offset field which can be used to specify an offset value for all IDs, which can be used to prevent such conflicts.

Question: I am having problems with what appears to be a corrupt database. How should I try to recover the model?

Common signs of a corrupted database include software aborts or unresponsiveness, or incorrect data being displayed. These problems are rare, but should especially be suspected when there have been previous aborts, particularly sudden ones such as powering down or rebooting the hardware during an operation.

To attempt to recover this model, first try to export the model into a FEMAP neutral file using the File/Export menu, and then import this file into a new, empty database file. If you are unable to do this, try to at least recover your analysis model using one of the file formats in the File/Export/Analysis Model menu. If you are licensed for the Professional Modeler option, you may also be able to create a Parasolid or ACIS file of your geometry for later import.

Above all, the best defense against a corrupt database is prevention. Maintain regular backup copies of your database files, and try to keep at least one copy off site from the computer itself in case of hardware or disk failure. Also, create regular copies of your database as a FEMAP neutral file, to aid in data recovery if needed.

Question: How can I change the units for my analysis model?

For existing models, the Tools/Convert Units menu allows you to explicitly set offset and scaling factor values to convert each type of numerical unit in your model. Most commonly, this menu is used in conjunction with the Load button, which allows you to read in a file of pre-defined conversion factors between common unit systems.

Normally, the analysis interprets the units of your model based on what is specified in your material properties. By default, material units are in the English (inches/pounds) system. By using the Load function when creating or modifying material data, you can also read in material definitions that are in SI or MKS units, and then modify their values as needed.

Question: How can I display the principal directions of my surface?

For meshing operations, a set of coordinate axes are drawn on the surfaces or solids to be meshed.

In addition, you can also display the parametric surface arrows for surfaces as follows: go to the View/Options menu, select Tools and View Style as a category, and then pick Curve and Surface Accuracy from the list of options on the left. Then, on the right hand side of the screen, set the Parametric Directions menu choice to Show Arrows. This option will display arrows showing the parametric orientation, which can then be used to determine the surface normal direction via the right-hand rule.

By default, there are three display division lines for each surface in the s parametric direction, and four division lines in the t direction. For a better understanding of the s versus t directions, you may also set a different number of display divisions in each direction. To do this, select the Modify/Update Other/Surface Divisions. Select one or more surfaces as desired, pick OK on the menu, and then specify the number of divisions for s and t in the subsequent dialog form.

Question: Is there a way to suppress small features in my model, to reduce its complexity?

For models created outside of MSC/NASTRAN for Windows, de-featuring of geometry is usually best done in its native CAD environment, prior to importing the model. Capabilities for de-featuring geometry in MSC/NASTRAN for Windows are limited to standard modeling operations. For the finite element model, however, there are two menu functions which can be useful in cleaning up your model:

1. The Mesh/Mesh Control/Feature Suppression menu contains numerous techniques for either automatically removing mesh small features subject to size constraints, or manually selecting edges, loops or surfaces for feature removal.

2. Small sliver elements created during the meshing process can be removed using the Mesh/Remesh/Clean up Slivers operation.

Question: Why is the maximum value that I see on my contour plot lower than the maximum nodal value that I get on my printout?

Contour plots of MSC/NATRAN analysis are created using nodal values, which are computed by averaging from individual values computed within elements. This averaging has an inherent effect of making your contour values somewhat conservative, e.g., less than the absolute peak value.

It is good engineering practice to use an adequate mesh density, particularly in regions of critical behavior, so that the results shown on a contour plot do not reflect high gradients (rates of change) within a single element. In situations where the peak values are important, it is also a good idea to use additional result displays such as a criteria plot, which display discrete results on individual color-coded elements.

The Smart Posting feature can also be used to prevent contour results from being averaged across element discontinuities such as different materials or properties.

Question: Can I copy my finite element model from one database to another?

Yes. By writing out an MSC/NASTRAN bulk data file or other analysis file format using the File/Export/Analysis Model function, you create a file that can be imported into another MSC/NASTRAN for Windows database using the File/Import function. Consult the table in Chapter 6 of the Users Guide for more information about what analysis data is supported in each output file format.

At the same time, we normally advise that you save your entire model, including geometry, to facilitate re-meshing and model editing in the future. This can best be accomplished by the export of a FEMAP neutral file from the File/Export menu, which can then be imported into a new database.

Question: When I move my model to a new directory folder, it no longer runs. Error # 9002.

MSC/NASTRAN for Windows requires database files to be in folders and pathnames that contain no spaces. Check each folder in the path name and check for the following:

1. Make sure there are no spaces in the folder name, substituting an underscore ("_") for spaces as needed.

2. Be sure that there are no national characters (such as !, @, # or other shifted-numerical keyboard characters) in your folders or filenames. Many of these characters are legal within Windows, but cannot be read properly by the software.

Question: How can I transfer a model from MSC/PATRAN to MSC/NASTRAN for Windows?

There are three ways to transfer data from MSC/PATRAN into MSC/NASTRAN for Windows:

1. For transferring geometry, use the IGES export capabilities of MSC/PATRAN (via its File/Export function) to create an IGES file which can be imported via the File/Import/Geometry function of MSC/NASTRAN for Windows.

2. To transfer a finite element model from MSC/PATRAN to MSC/NASTRAN for Windows, you can import an MSC/PATRAN Neutral File using the File/Import/Analysis Model menu.

3. For MSC/PATRAN installations which support MSC/NASTRAN, you may also generate an MSC/NASTRAN bulk data (.bdf) file and then import it into MSC/NASTRAN for Windows via the File/Import/Analysis Model menu. This file supports a broader range of analysis data and loading conditions than the MSC/PATRAN neutral file.

Question: The MSC/NASTRAN for Windows scratch file is getting too large for my disk. Can I allocate it to a different disk?

Use the File/Preferences menu to change this allocation. Select the Database button on the left side of this menu, then in the dialog that appears, set a new directory pathname for the scratch file in the options in the lower right hand corner.

Question: Do contour plots average results values across different materials?

The Smart Posting feature, when turned on, will prevent contour plots from being averaged across element discontinuities such as different materials or element properties.

Normally, contour plots are created from nodal values that are averaged across all elements sharing these nodes, even when they use different materials. To correctly reflect the discontinuity across a boundary between elements with different material properties, either use the Smart Posting feature, or consider using a criteria plot (which shows discrete values by color-coding individual elements), which displays un-averaged results on an element-by-element basis for the entire model.

Question: Can I perform MSC/NASTRAN for Windows modeling on one machine, and the solution on another?

MSC/NASTRAN for Windows is designed as an integrated system, where analyses are submitted directly from the modeling environment. However, since access to MSC/NASTRAN for Windows is controlled by a hardware security device, it is perfectly OK to maintain copies of the program on two machines, and switch the security device and database files as needed for analysis on a separate machine.

Question: Is there a way to read nodal forces from a text file into MSC/NASTRAN for Windows?

Yes. Create a text file with comma-separated values, with one record per node, in the following format:

NODEID,XVAL,YVAL,ZVAL

Where NODEID is the ID of the node, and XVAL, YVAL and ZVAL are the component force values.

Next, read in this file as an analysis result set using the File/Import/Analysis Results menu. Select the Comma Separated option, and then select the file using the subsequent file browser. Then, in the following menu, deselect First Row is Title, select Vectors are XYZ data, and make sure that the Nodal and First Column Contains IDs options are both set on.

After you have read in this file, create the load by selecting the Model/Load/From Output menu. Select Force, and then select the output set you have just created. Finally, select force components as desired, and pick OK to assign them to your model.

Once these forces are read in, and boundary conditions assigned, you can then use the Free-Body Diagram feature for graphical verification of these loads.