FEMAP

User Guide

Version 11.3

Proprietary and Restricted Rights Notice

Siemens PLM

Web: http://www.femap.com

Customer Support

Phone: (714) 952-5444, (800) 955-0000 (In US & Canada)

Web: http://support.ugs.com

Conventions

This manual uses different fonts to highlight command names or input that you must type.

`a:setup`	Shows text that you should type.
OK, Cancel	Shows a command name or text that you will see in a dialog box.

Throughout this manual, you will see references to Windows. Windows refers to Microsoft® Windows 7, Windows 8, Windows 8.1, and Windows 10 (64-bit versions only). You will need one of these operating environments to run FEMAP for the PC. This manual assumes that you are familiar with the general use of the operating environment. If you are not, you can refer to the Windows User's Guide for additional assistance.

Similarly, throughout the manual all references to FEMAP, refer to the latest version of our software.

Table of Contents

6. Element Reference

7. Translation Tables for Analysis Programs

8. Analysis Program Interfaces

1 Introduction

FEMAP is finite element modeling and post-processing software that allows you to perform engineering analyses both quickly and confidently. FEMAP provides the capability to develop sophisticated analyses of stress, temperature, and dynamic performance directly on the desktop. With easy access to CAD and office automation tools, productivity is dramatically improved compared to traditional approaches.

FEMAP automatically provides the integration that is necessary to link all aspects of your analysis. FEMAP can be used to create geometry, or you can import CAD geometry. FEMAP provides powerful tools for meshing geometry, as well as applying loads and boundary conditions. You may then use FEMAP to export an input file to over 20 finite element codes. FEMAP can also read the results from the solver program. Once results are obtained in FEMAP, a wide variety of tools are available for visualizing and reporting on your results.

Geometry

FEMAP can directly import geometry from your CAD or design system. In fact, FEMAP can directly import a solid model from any ACIS-based or Parasolid-based modeling package. If your modeling package does not use either of these geometry engines, you can use the FEMAP IGES or STEP reader. If you are using I-DEAS, you can bring a single part into FEMAP by exporting a Viewer XML (IDI) file from I-DEAS. These files can be read and then stitched together to form a solid. This typically requires using one command.

If you do not have CAD geometry, you can create geometry directly in FEMAP using powerful wireframe and solid modeling tools. Solid modeling directly in FEMAP uses the powerful Parasolid modeling engine. You can build or modify solid models using the Parasolid engine, and then export the geometry out of FEMAP. This is very convenient if you need to export geometry to CAD packages that are Parasolid-based.

Finite Element Modeling

Regardless of the origin of your geometry, you can use FEMAP to create a complete finite element model. Meshes can be created by many methods ranging from manual creation, to mapped meshing between keypoints, to fully automatic meshing of curves, surfaces and solids. FEMAP can even work with your existing analysis models. You can import and manipulate these models using the interfaces to any of the supported analysis programs.

Appropriate materials and section properties can be created or assigned from FEMAP libraries. Many types of constraint and loading conditions can be applied to represent the design environment. You can apply loads/constraints directly on finite element entities (nodes and elements), or you can apply them to geometry. FEMAP will automatically convert geometric conditions to nodal/elemental values upon translation to your solver program. You may even convert these loads before translation to convince yourself that the loading conditions are appropriate for your model.

Checking Your Model

At every step of the modeling process, you receive graphical verification of your progress. You need not worry about making a mistake because FEMAP contains a multi-level undo and redo capability.

FEMAP also provides extensive tools for checking your model before you analyze it to give you the confidence that you have properly modeled your part. It constantly examines input to prevent errors in the model, and provides immediate visual feedback. FEMAP also provides a comprehensive set of tools to evaluate your finite element model and identify errors that are often not obvious. For example, FEMAP can check for coincident geometry, find improper connections, estimate mass and inertia, evaluate your constraint conditions, and sum your loading conditions. Each of these methods can be used to identify and eliminate potential errors, saving you considerable time and money.

Analyzing Your Model

When your model is complete, FEMAP provides interface to over 20 popular programs to perform finite element analysis. You can even import a model from one analysis program and automatically convert it to the format for a different analysis program.

The NX Nastran for FEMAP solver is a general finite element analysis program for structural and thermal analysis that is integrated with FEMAP.

Post-processing

After your analysis, FEMAP provides both powerful visualization tools that enable you to quickly interpret results, and numerical tools to search, report, and perform further calculations using these results. Deformation plots, contour plots, animations, and XY plots of *Data Series* are just some of the post-processing tools available to the FEMAP user. FEMAP supports OpenGL, which provides even more capability for post-processing, including dynamic visualization of contours through solid parts. You can dynamically rotate solid contoured models with one push of your mouse button. Section cuts and isosurfaces can be viewed dynamically by simply moving your cursor.

Documenting Results

Documentation is also a very important factor with any analysis. FEMAP obviously provides direct, high quality printing and plotting of both graphics and text. Frequently, however, graphics or text must be incorporated into a larger report or presentation. FEMAP can export both graphics and text to non-engineering programs with a simple Windows Cut command. You can easily export pictures to popular programs such as Microsoft Word, Microsoft Power Point, and Adobe FrameMaker. You can export to spreadsheets, databases, word processors, desktop publishing software, and paint and illustration programs. These links enable you to create and publish a complete report or presentation, all electronically, right on your desktop.

With support for AVI files, you can even include an animation directly in your Power Point Presentation or Word document. FEMAP also supports VRML and JPEG format so anyone can easily view results with standard viewers.

FEMAP Documentation

FEMAP comes with a set of three printed manuals: *FEMAP Examples*, the *FEMAP User Guide*, and the *FEMAP Commands* reference manual.

The FEMAP on-line help includes the contents of these manuals, as well as several additional books. The complete set includes:

- *FEMAP Examples*: Step-by-step examples for new users.
- *FEMAP User Guide*: General information on how to use FEMAP, including an overview of the finite element modeling process. Also contains reference information for the FEMAP analysis program and geometry interfaces.
- *FEMAP Commands*: Detailed information on how to use FEMAP commands.
- *FEMAP API Reference:* Information on how to write your own applications that work with FEMAP.
- *What's New:* New features for this release.

When NX Nastran for FEMAP is installed, on-line help includes all of the above, as well as a full set of current NX Nastran documentation, to assist you during the solving portion of the analysis process.

Product Configurations

To best address the needs of our customers, FEMAP is available in two configurations: FEMAP and NX Nastran for FEMAP. Each configuration contains a license of FEMAP, giving you full access to all of the powerful modeling and post-processing capabilities of FEMAP. NX Nastran for FEMAP also includes the industry standard Nastran Finite Element Analysis solver to provide you a total analysis solution.

FEMAP

FEMAP includes automatic and manual meshing, automatic generation of beam cross section properties, support for a wide variety of material data, loading conditions, and analysis programs. FEMAP also includes automatic contact detection, advanced post-processing features, and robust solid and surface modeling using the Parasolid geometry engine.

FEMAP contains the Parasolid solid modeling engine. Parasolid is a solid modeling engine developed by UGS Corporation, and is the underlying modeling engine in many CAD and solid modeling engines such as Solid Edge, Unigraphics, and SolidWorks. FEMAP allows you to use powerful Parasolid-based geometry tools contained in FEMAP to create your own complex 3-D models from scratch. These 3-D models can be used to validate structural integrity inside of FEMAP, then exported out of FEMAP and imported into any Parasolid-based CAD systems for further manipulation, drawing creation, or incorporation into large assemblies of parts.

The ACIS-to-Parasolid geometry converter in FEMAP provides the ability to import solid models created with the ACIS Geometry Engine. ACIS is the solid modeling engine developed by Spatial Technology, Inc., and is used by several popular CAD systems including AutoCAD. If you frequently receive solid CAD data from ACIS-based CAD and solid-modeling systems, the ACIS geometry can be imported into FEMAP, will be converted to Parasolid automatically, modified inside of FEMAP, then used in the creation of effective FEA models. Parasolid geometry from FEMAP can also be exported out in ACIS format for use with ACIS-based CAD systems.

Finally, FEMAP includes direct interfaces to major CAD programs such as I-DEAS, CATIA, PRO/Engineer, Solid Edge (Parasolid), Unigraphics (Parasolid), and VDA, as well as the ability to both import and export geometry in the industry standard IGES or STEP formats.

FEMAP is the ideal solution for the analysts who receive CAD data from an outside source as well as create their own. The ability to import Parasolid, ACIS, IGES, and STEP files covers a wide variety of CAD systems. To idealize thin structures created as solids, FEMAP even provides excellent automatic and semi-automatic mid-planing capability. Therefore, you can import a thin solid from a CAD system, create a mid-plane surface representation of the part, and then mesh these surfaces with plates.

Leading firms recognize that it is unlikely a single analysis technology will meet all of their requirements. Moreover, by integrating multiple analysis technologies in a single modeling and visualization environment, they can make better designs faster.

NX Nastran for FEMAP

NX Nastran for FEMAP combines the power of the industry standard Nastran solver with the equally powerful modeling and post-processing capabilities of FEMAP.

NX Nastran for FEMAP currently supports:

- statics analysis solves for linear, static stress, and deflection results when thermo-mechanical loads are present.
- dynamic (normal modes) solves for natural frequencies and mode shapes of either restrained or free-free structures.
- advanced dynamics capabilities such as transient response, frequency response, response spectrum analysis, random response,
- nonlinear static and transient analysis lets you handle large deformations and material nonlinearity.
- both steady-state and transient heat transfer analysis solves for temperatures due to convection, conduction, heat generation and radiation.

- linear buckling analysis
- design optimization helps you to find more efficient design solutions
- advanced nonlinear capabilities including surface-to-surface contact using NX Nastran Solution 601 and explicit transient dynamics using NX Nastran Solution 701.

Getting Started

3

Welcome to FEMAP! This section will help you to setup your computer so that you can immediately begin to explore the many capabilities of FEMAP. Before you start however, take a few minutes to do the following:

1. Read the FEMAP License Agreement which was included with your CD. It limits how you may use this software on your computer. Typically, you may only use FEMAP on one computer, for use by one individual at a time.

2. Fill out and fax the License Registration sheet which was included with your CD. Returning this document will insure that you will receive telephone support if you need it, and that you will be notified of future enhancements and corrections to FEMAP.

3. If you used a previous version of FEMAP, see Section D, "Converting Old Models" for information pertaining to conversion of old databases to this version of the software.

This section contains information specific to getting started on a PC, which includes 64-bit versions for Windows 7, Windows 8, Windows 8.1, and Windows 10. The FEMAP installation DVD contains only the 64-bit version of FEMAP, as a 32-bit version is no longer available.

Note: You MUST be logged in with Administrator privileges when installing FEMAP for the installation to proceed.

3.1 Hardware Requirements

There are no special hardware requirements for FEMAP beyond those imposed by Windows. There are many types of hardware that will allow you to use FEMAP. Proper choice of hardware however, can often make the difference between frustration and productivity. Here are a few suggestions:

Memory, RAM:

You will need at least 128 Mbytes of RAM to run FEMAP and the Parasolid solid modeling engine, which is the default. Obviously, the more amount of RAM the better. Adding RAM can be one of the most cost effective means of increasing performance.

Note: If using the “Standard” geometry Engine in FEMAP, you can actually run with as little as 32 Mbytes of RAM. This is not a recommended configuration.

Memory, (Hard Disk):

Required hard disk space is very difficult to estimate, but in general you will never have enough. Analysis results will be the main driver of any disk space requirement. Models are typically relatively small. A model with 1000 nodes and 1000 elements would typically be less than 1 Mbyte in size. Output from an analysis of that model however could be 5 Mbytes, 10 Mbytes or even much larger, depending on the output you request. To estimate total disk space, you need to first estimate how many models you will have on-line simultaneously, the approximate size of those models, and the type of output you will request.

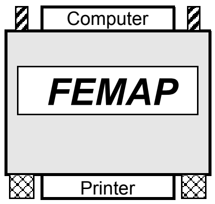
Graphics Boards:

While, standard graphics adapters may work very well with FEMAP, having a specialized board with support for OpenGL will provide increased graphical performance when dynamically rotating large, complex models. They also usually provide higher resolution and more colors, which make graphics easier to see and more realistic. Also, in order to use the “Performance Graphics” option, a graphics card which supports OpenGL 4.2 is required.

3.2 Installation - Stand Alone

This section describes the procedure that you should follow to install the stand alone (security device) version of FEMAP on your PC.

3.2.1 Security Device



In order to run the Stand Alone (Security Device) version of FEMAP a Rainbow SuperPro USB Port or Parallel Port (pictured on left) dongle is required. In order for your PC to be able to see the dongle, a driver must first be installed. Installation of the driver requires Administrator privileges for your PC. During installation, if the current user has Administrator privileges, the installation program will automatically prompt for installation of this driver.

If the installer does not have Administrator privileges, someone with Administrator privileges may log in and install the driver manually. The driver installation program can be found in the SentinalDriver directory of the FEMAP DVD. On any supported Windows platform, run DVD\SentinalDriver\SSD759.exe. It is highly recommended that you do not have any security devices attached to your computer while you are installing the driver. Once the driver has been installed, you can plug a USB security device directly into an open USB port and it should be recognized. For the Parallel Port security device, it is highly recommended that you shut your computer down and turn it off before installing the security device. After it is installed, turn the computer on begin using FEMAP.

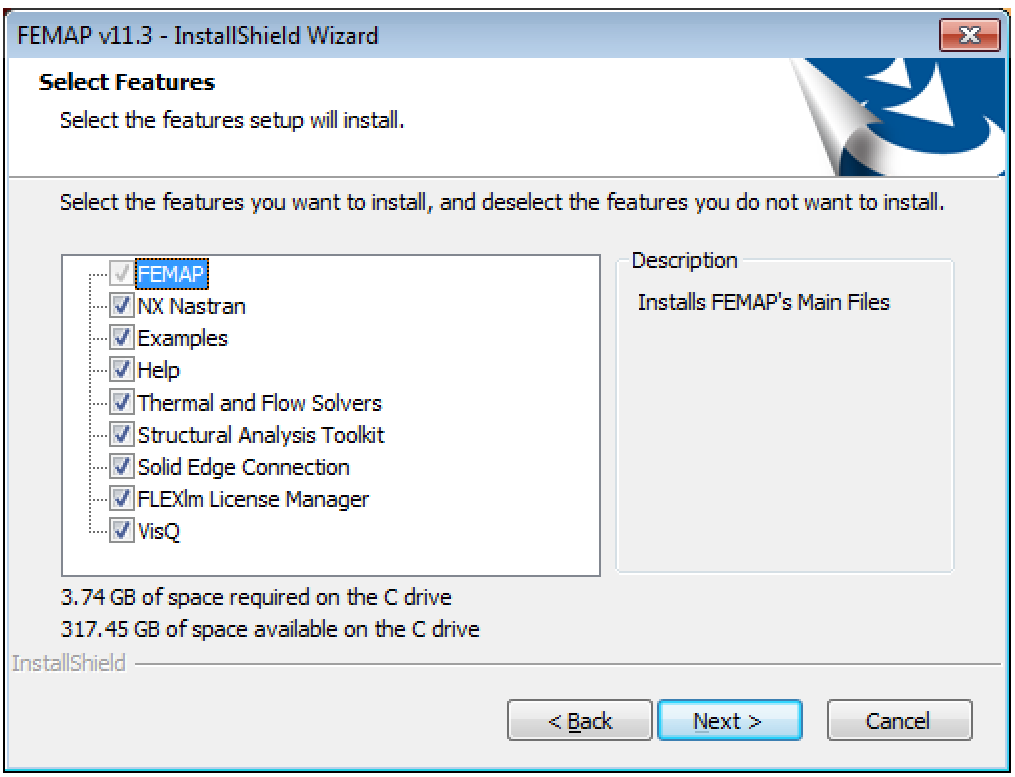
3.2.2 Setup Program Execution

Windows 7/8/8.1/10

1. Log in to your computer as Administrator. As detailed above, this will make installation of the driver required to talk to the FEMAP dongle possible.

2. Insert the FEMAP DVD into the drive. The setup should automatically begin within a few seconds. If it does not, manually run the SETUP.EXE program in the root directory of the FEMAP DVD

Once setup is running you will see a license agreement. Assuming that you agree with the license agreement, choose “I accept the terms of the license agreement” and press *Next* to continue and select the directory where you would like to have the FEMAP program files installed. You will be prompted for the selection of additional FEMAP options, please choose any optional modules and components that you wish to have installed.

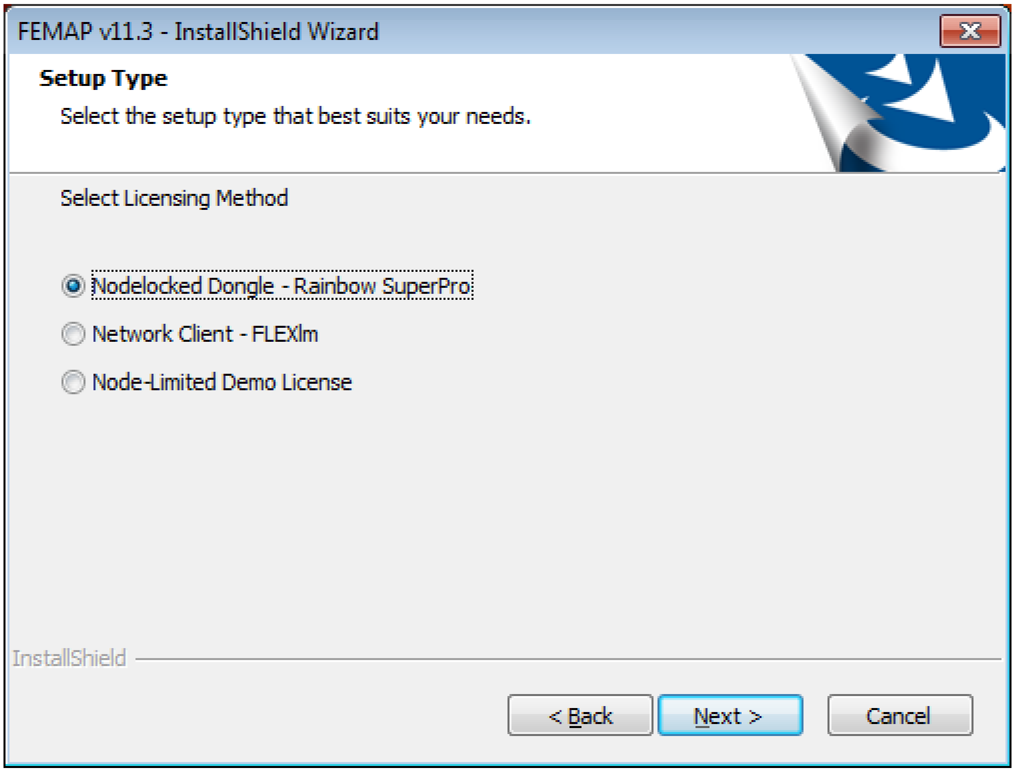


Notice that the installation will tell you the amount of disk space required for the chosen options to be installed and how much space is available on the drive where FEMAP will be installed.

Note: If you plan on licensing FEMAP with a dongle (security key), not a network license, then you will probably want to UNCHECK the *FLEXlm License Manager* option as it is not used by the dongle.

Next, a dialog box allows you to *Select FEMAP GUI Language*. Select from *English*, *German, Simplified Chinese, Traditional Chinese*, or *Japanese* then click *Next*.

You will now be asked which type of installation to perform. Choose Nodelocked Dongle as the licensing method



Setup Type	Description
Nodelocked Dongle - Rainbow SuperPro	Installs FEMAP for use with a Rainbow Parallel Port or USB dongle. If you have the dongle version of FEMAP, choose this setup type.
Network Client - FLEXlm	Installs the Network Client version of FEMAP. This setup is for use where FEMAP is licensed via the FLEXlm license management software. With the Network Client version of FEMAP, one machine on your network will be designated as the license server. The following “Network License Server” setup will have to be run on that machine.
Node-Limited Demo License	Installs the 300-Node demonstration version of FEMAP. This version requires no licensing, but is limited to very small models. It is intended for new users to try FEMAP and all its options.

After choosing Nodelocked Dongle and pressing *Next*, the program will be installed and then a driver required for the dongle will automatically be installed. Finally, if you are installing FEMAP with the NX Nastran option you will be prompted to specify a “scratch” directory for the solver. You will need to have read/write access to this directory to be able to properly use NX Nastran.

3.2.3 Upgrading Your Security Device

FEMAP dongles are shipped good for 30 days from the first time they are run. In order to remove the time limit from your new FEMAP dongle, or upgrade an older dongle or network license, you must contact the Siemens Product Lifecycle Management Software Inc. Global Technical Access Center (GTAC). Upgrade codes and updated license files are now available via Siemens Product Lifecycle Management Software Inc. GTAC (Global Technical

Access Center) WebKey system available on-line at https://plmapps.industrysoftware.automation.siemens.com/webkey/ FEMAP customers can use WebKey for both licensing support and product technical support.

3.2.3.1 *Obtaining a Webkey Account from Siemens Product Lifecycle Management Software Inc.*

To request a WebKey account, you can attempt to use the Help, Technical Support, Request Webkey Account command or simply access the web page using this URL:

https://plmapps.industrysoftware.automation.siemens.com/webkey/

Then provide the following information:

- Your Installation ID
- WebKey Access Code

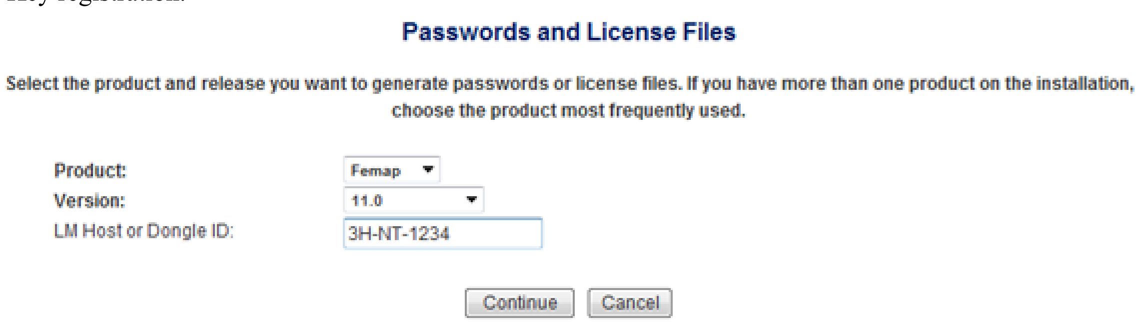
Your Installation ID is directly under the “Sold To” information on your shipping order. For dongle-based FEMAP customers, your WebKey Access code is the unique portion of your FEMAP serial number, i.e. 3H-NT-1234, which is displayed in your current FEMAP in the Help - About dialog box, for this license as 1000-3H-NT-1234, with the version information at the beginning of the serial number removed. If you have any problems determining your Installation ID, FEMAP Serial Number, or have trouble getting a WebKey account, please contact:

Trish McNamara - trish.mcnamara@siemans.com - 610-458-6508, or

Mark Sherman - sherman.mark@siemans.com - 610-458-6502

3.2.3.2 *Obtaining Upgrade Codes or a new License File*

1. Via the Web, using your WebKey Account -Upgrade codes or an updated license file can be e-mailed to you from the Customer Support (GTAC) web site http://www.plm.automation.siemens.com/en_us/support/gtac. In the *Explore GTAC* section, expand “License Management” and select “Current Licenses”. If prompted, enter WebKey and password. Click *Passwords and License Files* link. Select “Femap” as the *Product* and set *Version* to the appropriate version (i.e., 11.1 or 11.0). For *LM Host or Dongle ID*, enter either the unique portion of you FEMAP serial number (3H-NT-1234 in this case) if using a dongle or fill in the Ethernet address of your FEMAP license server if using FLEXlm network licenses. Your license or access codes will be e-mailed to the address supplied during WebKey registration.



2. Via the Phone - You can call GTAC at 714-952-5444 (US and Canada residents may use 800-955-0000) and enter option 1, 1, for your CSR or option 1, 2, for Software Product Delivery (SPD). You should then request a copy of the license upgrade for a specific Installation ID and serial number or Ethernet Address.

For dongle versions of FEMAP, the information returned to you to upgrade the dongle will be in the form of two case insensitive alpha numeric codes. They will appear something like:

Access Code 1: 08aeca3f0f52639179

Access Code 2: 362ff63c3426d943

Use the Help, About command, then click the Security button. Cut and paste (to avoid errors) or type these two codes in to the appropriate fields and press OK. The FEMAP dongle is an EPROM, and these codes are used to update the memory of the dongle. Once these codes have been entered, you will never need to enter them again, with changes made to the memory of the dongle, they will either be useless, or simply write the same thing to memory again.

3.3 Network Installation - PC

The “Network Client” version of FEMAP utilizes the FLEXlm License Manager software from Flexera Software. This licensing approach requires some software to be installed on a server machine and other software to be installed on one or more clients. The clients then request and obtain licenses from the server. In a simple situation, both the client and server could be the same computer, but more likely they are different systems connected by a network.

3.3.1 Obtaining a License File

License files are obtained through the same procedure as defined above for getting the upgrade codes for a dongle license. Call GTAC, or use your WebKey account to request your FEMAP license file. The only difference in Network Licensed FEMAP is that you need to enter the LMHostID (Ethernet Address) of your license server when prompted instead of the FEMAP Serial Number. When you receive your license file information, you need to extract just the valid FLEXlm license entries, and copy them into a file called “license.dat”. Please make sure that your license.dat looks something like the one show below. For FEMAP, you will have one SERVER line, one DAEMON line, and one or more FEATURE lines depending on how many options you have purchased with your FEMAP.

A couple of things to make sure of:

1. Make sure that the entry immediately following the word “SERVER” is the name of the license server where you are installing the license server software. If it is a temporary name, i.e. ANY, or THISHOST, change it to the correct machine name. This is one of the two things in the license file that you can change.

2. Make sure that the third entry on the SERVER line matches the LMHostID of license server. This number is the key to the whole license file. If this does not match the LMHostID of the license server, then the licensing will not work.

3. The “DAEMON esplmd” line calls out the actual programs that hands out FEMAP licenses. If you have installed all the license server pieces in the same directory, it is fine as is. If the esplmd.exe program is not in the same directory as LMTOOLS.EXE, you will have to edit this line to tell LMTOOLS.EXE where to find it. This is the other part of a license file that you can change.

```
SERVER PHLF10 00095b8e20ef
DAEMON esplmd
FEATURE femap esplmd 9.90 25-nov-2013 1 BFAD29839CA1 \
        VENDOR_STRING=990-4H-NT-DEV0001802070000 SIGN2="006C 1900 \
        0B32 DD61 C04C FA5A ECE7 CF00 47E8 DB2B 4387 0D1C BC34 91BF \
        8186"
FEATURE femapnx_nas_basic_fep esplmd 9.90 25-nov-2013 1 E21B2CF151CE \
        VENDOR_STRING=990-4H-NT-DEV0001802070000 SIGN2="00D8 E0DB \
        E270 E582 D98A 143D 3C5E B700 82D6 0CD9 1D9D 6CD7 5A91 BAF0 \
        428E"
FEATURE femapnx_nas_nonlin_fep esplmd 9.90 25-nov-2013 1 CDB61D1114C2 \
        VENDOR_STRING=990-4H-NT-DEV0001802070000 SIGN2="00C2 C156 \
        A207 51D8 3238 7B87 9337 3000 C44A 8243 C861 9096 8EDF 7A6C \
        AB39"
FEATURE femapnx_nas_dyn_fep esplmd 9.90 25-nov-2013 1 DA870A4F7A69 \
        VENDOR_STRING=990-4H-NT-DEV0001802070000 SIGN2="0055 A931 \
        3AE1 8E9F 9236 BA1F 22AC 6400 07BA D084 A8F7 3404 4A19 DFB3 \
        FEA0"
```

3.3.2 License Server

This section provides instructions on installing the network license manager and configuring your server.

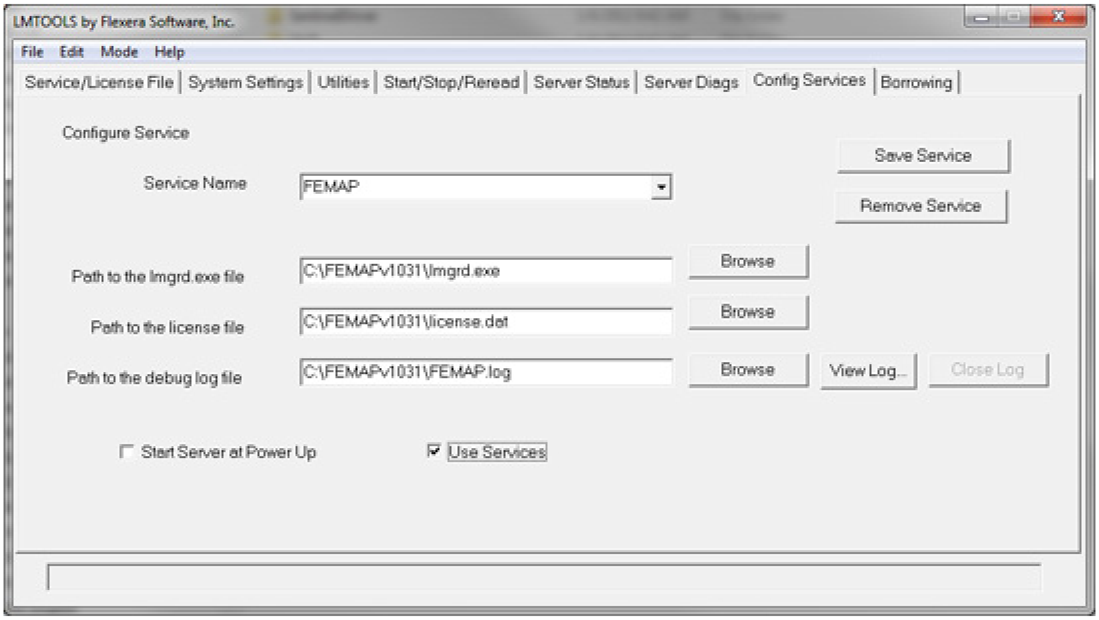
3.3.2.1 Installing the FLEXlm License Manager

To begin the server installation, simply insert the FEMAP CD and allow it to AutoRun, or choose setup from the CD. FEMAP will ask which “features” should be installed. If you only want to install the license server, then UNCHECK all the options except “FLEXlm License Manager”. Once FEMAP has installed the software, copy your license file (usually called “license.dat”) to the same directory where you installed the license server components.

3.3.2.2 Configuring the FLEXlm License Manager

You can run the LMTOOLS program from the FEMAP entry on your start -> All Programs - >FEMAP v11.2 -> FLEXlm License Manager, or manually run LMTOOLS.EXE from its installed directory.

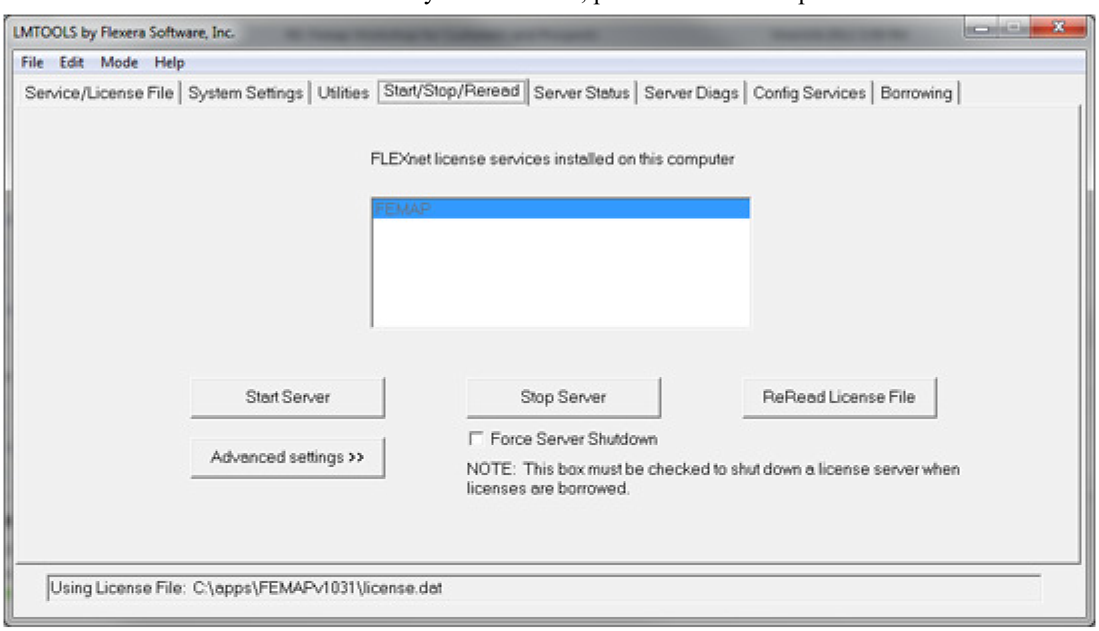
Once LMTOOLS is running, select the “Config Services” Tab.



Fill in a Service Name, specify a path to the lmgrd.exe file (a required FLEXlm component) that can be found in the installation directory, and specify the path to the license file. Finally, check the “Use Services” option, and then the “Start Server at Power Up”. Press the “Save Service” button.

Answer “Yes” to the question: “Would you like to save the settings for the service: FEMAP?”

You must start the license server manually the first time, press the “Start/Stop/Reread” tab.



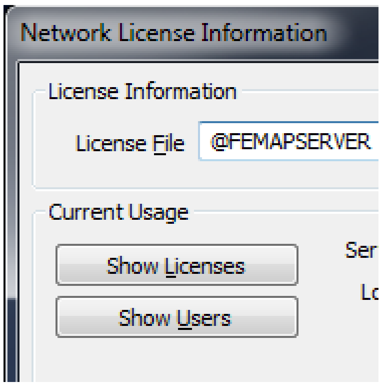
Select the FEMAP service that you just created, and press the “Start Server” button. At this point FLEXlm will be handing out FEMAP licenses on your network. To verify that everything is working fine from the license server standpoint, press the “Server Status” tab.

Press the “Perform Status Enquiry” button and the text window will be filled with status information about your FLEXlm license server. In the text window you will find information about how many licenses are available, and once user start checking out licenses, how many are in use.

3.3.3 Configuring Network Client Machines

Once your network license server is up and running, configuring FEMAP Network Client machines is very easy. Make sure that FEMAP is installed on the local machine using the “Network Client” setup type. To configure client machines to access the network license:

You have two options for telling network client machines how to find licenses on the license server:



1. Place a copy of the “license.dat” file in the FEMAP directory on the client machine. FEMAP will extract the name of the license server from the license file, and check out a license and run. The only drawback to this approach is that you must remember to update every copy of the license file when you receive a new one from Siemens Product Lifecycle Management Software Inc. (updates, licensing changes, etc.). To avoid this problem, you can type in the full network path to the License File in the “License File” field used below for HostName/IP Address location of the license server.

2. Tell FEMAP the name or IP address of the License Server.

a. Start FEMAP

b. Go to Help - About - Security

c. In the “License File” field, enter the name of the license server, preceded by an ampersand. In the example below, FEMAP is told to check out licenses from a network machine named PLSRV2:

d. In order for this machine name approach to work, the client computer must be able to see the license server computer via TCP/IP networking. To verify this, you can open a Command Prompt and ping the license server. In this case, one would type “ping PLSRV2”. The ping command will let you know if it can talk to the machine name indicated. If the client computer cannot find the license server by its name, you can also enter the IP address of the license server, preceded by an ampersand and licensing should also work.

3.3.4 Monitoring Network Usage

In a multi-user environment, sometimes you will not be able to get a license simply because all available licenses are in use. You can find out who is using licenses, which computers they are using and when they started their license simply by going to *Help, About*, and pressing the *Security* button. At the bottom of the dialog box you will find information that will give you this information.

If you fail to get a license because none are available, you will not be able to work in FEMAP. You do not however, have to leave FEMAP. You can simply stay there and periodically try a command. Whenever a license becomes available it will be assigned to you and your command will succeed. If there are still no licenses available, you will simply get a message that says try again later.

3.3.5 Copying FEMAP from one machine to another

In previous versions, the FEMAP directory created from a proper installation could simply be copied from one machine to another, then with the proper licensing, could be run on the new machine. For FEMAP 11 and above, there are some additional steps which must be done in order for a copied version of FEMAP to be able to run.

Note: You must have Administrator privileges on the machine FEMAP is being copied to in order to complete these additional steps.

First, you will need to run “vcredist_x86.exe”, then also run a 64-bit version of the executable called “vcredist_x64.exe”. You need to run both because FEMAP still uses some 32-bit applications. Additional redistributable executables may need to be run on certain operating systems.

Next, you must start FEMAP using the “Run as Administrator” option available by right-mouse clicking on the “femap.exe” file. Running FEMAP from an Administrator account is typically not sufficient to properly write to the registry.

Finally, using a DOS prompt, navigate to the FEMAP install directory, then type:

femap/register

...then press *Enter*. This will fully resister the application on this machine. This only needs to be done once, then FEMAP should run normally and API capabilities will be available.

3.4 Starting FEMAP

There are several command line options to launch FEMAP. The simplest method to launch FEMAP is to create a shortcut for FEMAP on your desktop and double-click the icon when you want to launch FEMAP. This will use the command line contained under the shortcut to launch FEMAP. You can modify this command line by right-clicking on the FEMAP icon, selecting properties, and changing the command line option on the shortcut.

The command line will contain the executable (and its path). After the femap.exe, there are several options which may be used to determine the mode in which FEMAP will operate. A list of these command line options are provided below.

```
c:\FEMAPv###\femap.exe [-R] [-NEU] [-NOSPL] [-D dxf_file] [-N neu_file]
[-PRG program_file] [-SE Solid Edge_File] [-L port] [-SAT sat_file]
[-XMT x_t file] [-SCA scale_factor] [-IGES iges_file] [model_file or ?]
```

where all of the arguments in [ ] are optional command line parameters. They are:

The remaining parameters can be specified in any order.

-R	Read Only Mode. With this option set, the *Save, Save As* and *Timed Save* commands are disabled. You will not be able to save changes to any model you access. All other commands remain active. Any changes you make will be made in the temporary scratch file, and will be lost when you exit FEMAP.
-NEU	Automatically writes a neutral file with the same name (just .NEU extension) as your .MOD file every time you save a model. In addition, when you open a model, if a neutral file exists with a newer date than the model, it will be read.
-NOSPL	Starts FEMAP without the splash screen.
-D dxf_file	This option automatically reads the specified DXF file when you start FEMAP. Make sure you leave at least one space between the two arguments.
-N neu_file	This option automatically reads the specified FEMAP neutral file when you start FEMAP.
-PRG program_file	This option allows you to run a specified FEMAP program file (*.PRO or *.PRG file) when FEMAP is started.
-SE Solid Edge_file	Automatically creates a new FEMAP file and calls the *File, Import Geometry* command to read the Solid Edge part file (*.prt file) or assembly file (*.asm file). When you use FEMAP with this command option, you will see the *Solid Model Read Options* dialog box, which will contain the title of the solid model file contained in the SAT file.

-L port

Specifies the parallel port where the FEMAP security device has been installed. This is not typically needed unless FEMAP has difficulty accessing the device. If you want to attach the security device to parallel port 1 (LPT1:), use -L 1, for parallel port 2 (LPT2:) use -L 2. If your system is non-standard, or uses some other parallel port convention, you can specify the actual parallel port address. For example, if your parallel port was at address 03BCH (hexadecimal), you would convert the address to a decimal value, in this case 956, and specify -L 956.

If you need to specify the -L option, you can change the default command line associated with the FEMAP icon on the Desktop by selecting *Properties.* First, right-click on the FEMAP icon. Then choose the *File, Properties* command (or press *Alt+Enter*). Move down to the command line option, and just add the appropriate -L options. From then on FEMAP will look for the security device on the specified port.

-SAT sat_file

Automatically creates a new FEMAP file and calls the *File, Import Geometry* command to read the ACIS solid model file *.SAT file [sat_file]. When you use FEMAP with this command option, you will see the *Solid Model Read Options* dialog box, which will contain the title of the solid model file contained in the SAT file.

-XMT xmt_file

Automatically creates a new FEMAP file and calls the *File, Import, Geometry* command to read the Parasolid solid model file *.X_T file [xmt_file]. When you use FEMAP with this command option, you will see the *Solid Model Read Options* dialog box which will contain the title of the solid model file contained in the X_T file.

-SCA scale_value

This option is used in conjunction with the -XMT and -SAT to specify a scale factor for the solid model. If this option is used, FEMAP will automatically import and scale the solid model. The *Solid Model Read Options* dialog box will not be shown.

-IGES iges_file

Automatically creates a new FEMAP file and calls the *File, Import, Geometry* command to read the file [iges_file]. When you use FEMAP with this command option, you will see the *IGES Read Options* dialog box, where you can specify options for reading the file.

- INI filename

Specify a specific femap.ini file to use. The femap.ini file contains specific options which can be used to customize many aspects of the program, such as a specific set of values for *File, Preferences*.

model_file

Normally FEMAP will start with a new, unnamed model. If model_file is the filename of an existing model however, FEMAP will start using that model. If the file does not exist, you will see an error message, and FEMAP will start a new model with that name.

?

If you add a question mark to the command line instead of specifying a model name, FEMAP will automatically display the standard file access dialog box and ask you for the name of the model that you want to use. If you want to begin a new model, just press *New Model* or the *Escape* key. When you want to work on an existing model, just choose it from the dialog box, or type its name.

You should never specify both the ? and model_file options.

3.4.1 Errors Starting FEMAP

Security Device Not Found

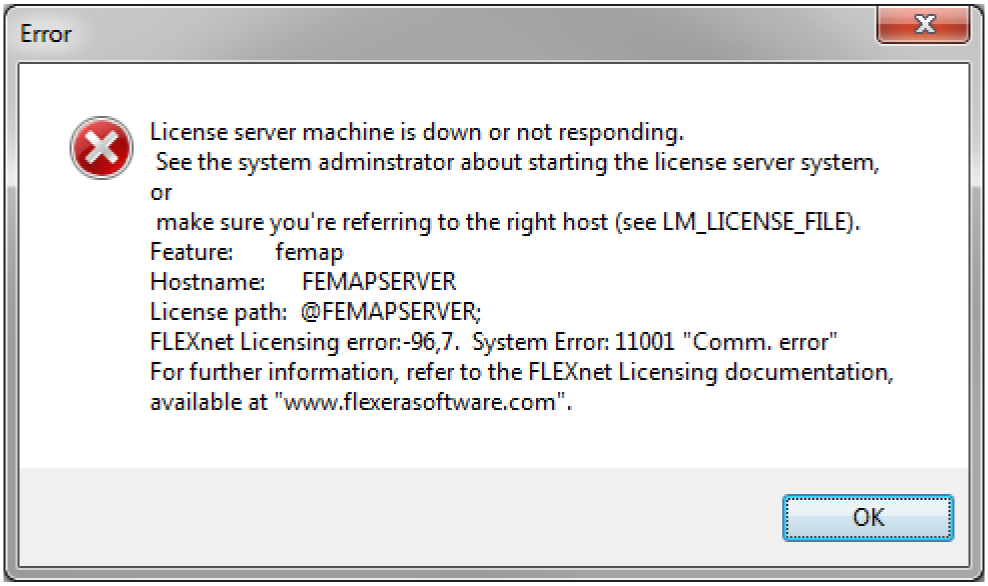
Symptom:

You see an error indicating that the security device cannot be found.

Resolution:

Go to Section 3.2.1, "Security Device", and confirm all steps have been followed. Try to run FEMAP again.

Choose Server or File



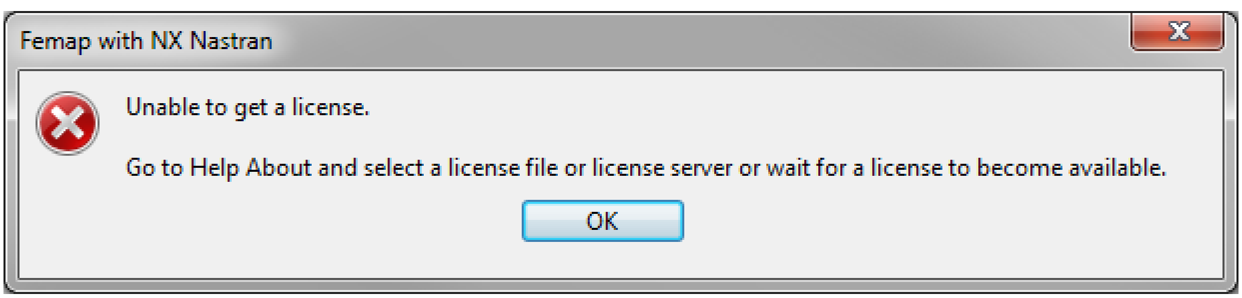
Symptom:

If you are attempting to start a network client and see the *Error* dialog box from FEMAP, FLEXlm cannot find a valid license file.

Resolution:

Press *Cancel* in this dialog box. Pick *Help, About, Security* to define the location of the license file, as instructed above in Section 3.3.3, "Configuring Network Client Machines"

Unable to get license error message



LM_LICENSE_FILE Environment variable message:

LM_LICENSE_FILE environment variable defined. It overrides all license file paths, and if it points to a license for a different product, it will cause this licensing failure. You must either redefine or remove this definition, or merge your license file with the one specified.

This error will ONLY occur when the environment variable LM_LICENSE_FILE has been set. For example, this environment variable may have been set by another application for licensing purposes. Be careful when removing or altering this environment variable as it may cause other applications to no longer function properly.

Other Error Messages

Symptom:

If you receive an “Unable to access {directory path}. Either this directory does not exist or you do not have proper permissions. Check the directory and your preferences” error or have any other difficulty starting FEMAP where abnormal termination occurs, you either do not have enough disk space, or your Windows TEMP is not set to a valid, accessible directory.

Resolution:

You may either change your Windows TEMP directory environment variable, or specify a path for the FEMAP scratch files (which default to the Windows TEMP directory set by the environment variable) to a valid directory.

This and all other FEMAP preferences are stored in a file called femap.ini that is typically located in the FEMAP executable directory. You will have to create this file or modify it to include the appropriate lines as shown below:

```
DISKMODELSCR=C:\FEMAP112
```

where `C:\FEMAP111` can be any valid path. The `DISKMODELSCR` parameter is case sensitive and must be defined exactly as above. Once you make these changes and FEMAP starts, you can use the *File, Preferences, Database* command to modify this path.

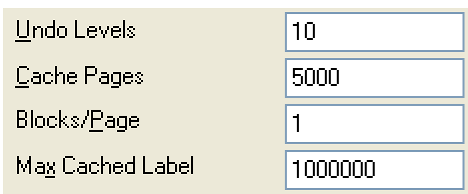
3.4.2 Improving Performance (RAM Management)

For version 9.3.1 and above, FEMAP determines the amount of available memory a machine and sets it to a default level automatically (20%).

FEMAP performance may improve on Windows personal workstations by modifying the default settings that FEMAP uses to manage RAM. To view or change these settings, use the *File, Preferences* command, then click the “Database” tab. For more details on this procedure, see Section 2.6.2.5, "Database" in *FEMAP Commands* manual.

3.4.2.1 Setting Guidelines for older versions of FEMAP (7.0 to 9.3)

To access the internal FEMAP memory management system, follow the procedure below:



1. Choose the *File, Preferences* command and click the *Database* tab.

2. Change *Cache Pages* and *Blocks/Page* in the dialog box.

3. *Max Cached Label* should be set to a number that is higher than any entity you will create in your model file. This sets aside a small portion of memory that stores all of the IDs in FEMAP.

4. Select *OK*.

The total amount of memory that FEMAP can allocate is the product of the *Cache Pages*, *Blocks/Page*, and 4096 bytes/block (i.e. the values shown, 12000 x 2 x 4096 will allow FEMAP to access just under 100 Mbytes of RAM) and use entity IDs up to 6,000,000.

Note: You should never allow FEMAP to allocate more than the physical memory of the machine. The internal memory management (swapping) in FEMAP will allow the program to run much faster than Windows memory swapping. Therefore, you should set the *Cache Pages* and *Blocks/Page* at a level which is comfortably below the physical memory of the machine. Also, to optimize performance, you should always increase *Cache Pages* (max 15000) to its limit before increasing *Blocks/Page*.

FEMAP Version 7.0 to 9.3

The following figures are provided as a starting point to improve performance.

Operating System	Installed RAM (Mb)	Cache Pages	Blocks/Page
Windows XP, 2000, Me	64	6000	1
	128	8000	2
	256	12000	3
	512	15000	5
	1000	15000	11

Actual performance will vary depending upon other concurrent applications and model specifics. Once again, it is best to increase *Cache Pages* to 15000 before increasing *Blocks/Page*.

Note: For best performance you should have enough physical RAM to load the entire model file into memory. For example, if you expect your model files to be a maximum of 100 Mb, then you would want FEMAP to allocate at least 100 Mb of memory. If you had 128 Mb of physical RAM, this would leave 28 Mb for Windows and other programs that may be running at the same time as FEMAP.

3.5 Licensing Conversion Methods

Please read this section very carefully before changing your licensing method. If you are going to convert your licensing method you MUST HAVE FEMAP AND NX NASTRAN CLOSED (not running) before you use the files described below.

You can change your licensing method (i.e., from using a security key to using a network license) using specific “batch” files located in the FEMAP directory. The files are named “go_licensing method”.bat and require minimum user input to change your licensing method. In general, the “go” batch files change your current “auth_###.dll” to use the appropriate licensing method (auth_licensing method.dll) and may create or alter some other required files. FEMAP will open a “command prompt” and let you know if the conversion of the auth_###.dll has been successful. The various “go” files are explained in greater detail below:

- go_apionly.bat - converts your current licensing method to the “API Only” version of FEMAP
- go_demo.bat - converts your current licensing method to the FEMAP Node-Limited Demonstration version.
- go_dongle.bat - converts your current licensing method to use a security key.
- go_network.bat - converts your current licensing method to use the FlexLM Network Client.

3.6 Release Management and Service Bulletins

Important information regarding current and future releases of FEMAP is often distributed electronically by Siemens PLM Software. To subscribe to the various mailing lists, please use the following link:

http://support.industrysoftware.automation.siemens.com/general/email.shtml

4 User Interface

This section describes the FEMAP user interface. It is divided into four major sections:

- The first section describes the overall graphical interface, as well as its relationship to FEMAP.
- The second section involves accessing commands in FEMAP. There are eight major methods of accessing commands: Main menu, Toolbars (Standard and Custom), Quick Access menu (right-mouse button in graphics window when Select Toolbar is not active), Shortcut keys, Status bar, The Select Toolbar (alternative gateway to many useful commands), Context Sensitive menus, and Dockable Pane icons.
- The third section describes common dialog boxes in FEMAP.
- The fourth section provides information on the FEMAP workplane and other tools.

4.1 Overview

This section provides an overview of the graphical user interface for FEMAP. Explanations of FEMAP's connections with Windows, as well as a general overview of the Windows which comprise the graphical user interface are provided. This section is divided into a brief description of the overall FEMAP interface, the FEMAP main window, the FEMAP *Messages* window, and the FEMAP Graphics window.

4.1.1 The FEMAP/Windows Team

FEMAP is a true Windows program. Therefore, if you have any experience running a Windows program, you will understand the FEMAP format. Careful implementation of Windows standards makes learning to use multiple Windows applications much easier since there are many similarities in the user interface.

The other distinct advantage of being a true Windows program is you can easily export information directly from FEMAP to other Windows programs. This is extremely useful when generating reports with graphical information such as color contours. You can simply use a *File, Picture, Copy* to generate a Metafile copy on the clipboard, and use an *Edit, Paste Special* in such programs as Microsoft Word to paste the picture into a Word document. You can then even scale this picture in Word since it is contained in Metafile format. Similarly, you can copy information from the FEMAP *Messages* windows to such programs as EXCEL for further manipulation of data.

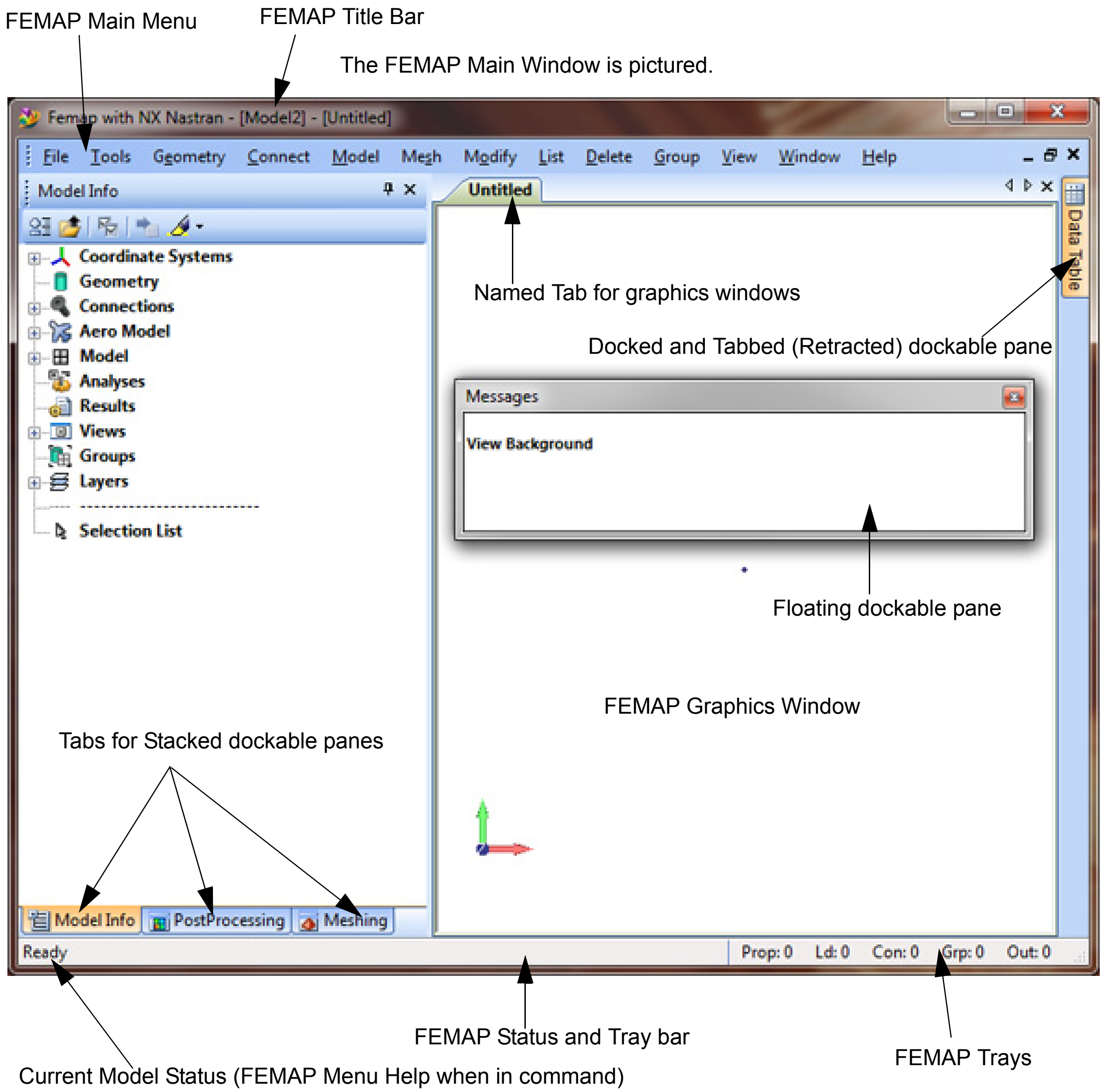
4.1.2 The FEMAP Windows

The two most basic visible parts of any Windows application are its window and the dialog boxes that it uses for input. Dialog boxes are typically only displayed when input is required, and then disappear. Conversely, windows usually remain visible, sometimes even floating or docked into position, to present text or graphical information until you decide to close, hide, or destroy them.

FEMAP uses three distinct types of windows: the FEMAP Main window (also referred to often as the Interface or Application window), multiple "tabbed" Graphics windows, and other "dockable pane" windows. The dockable panes include the *Messages* window, *Entity Editor*, *Data Table*, and *Model Info* tree. Other specialty FEMAP features appear in dockable panes such the *Stress Wizard* and the *Analysis Monitor*.

The figure below notes the location of the all three window types, some of the dockable panes in optional configurations, as well as other features inside of these windows. A description of each of these window types is provided.

Note: No toolbars are show in this view to make it easier to view the different types and configurations of FEMAP windows. Standard and Custom Toolbars will be covered later in this section.



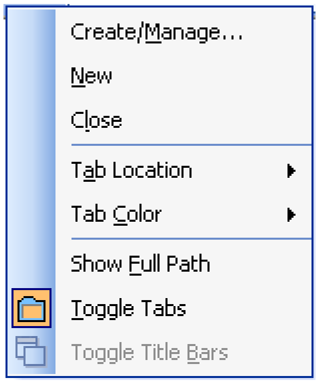
4.1.2.1 The FEMAP Main Window

The FEMAP main window is the "parent" or "application window" for FEMAP. When you begin FEMAP, this window will occupy your entire screen. It can be resized and moved using the standard Windows methods, but it cannot be closed or destroyed (unless you exit FEMAP).

The main window defines the "FEMAP Desktop", the portion of the screen that will be used by starting FEMAP. Although you can manually move FEMAP's other windows outside of the boundaries of the main window, their default sizing and position will always be inside this window. Similarly, if you minimize this window turning it into an icon, all other FEMAP windows (Graphics and Dockable Panes) will disappear. They will automatically reappear when the icon is restored to its original form or maximized to full screen.

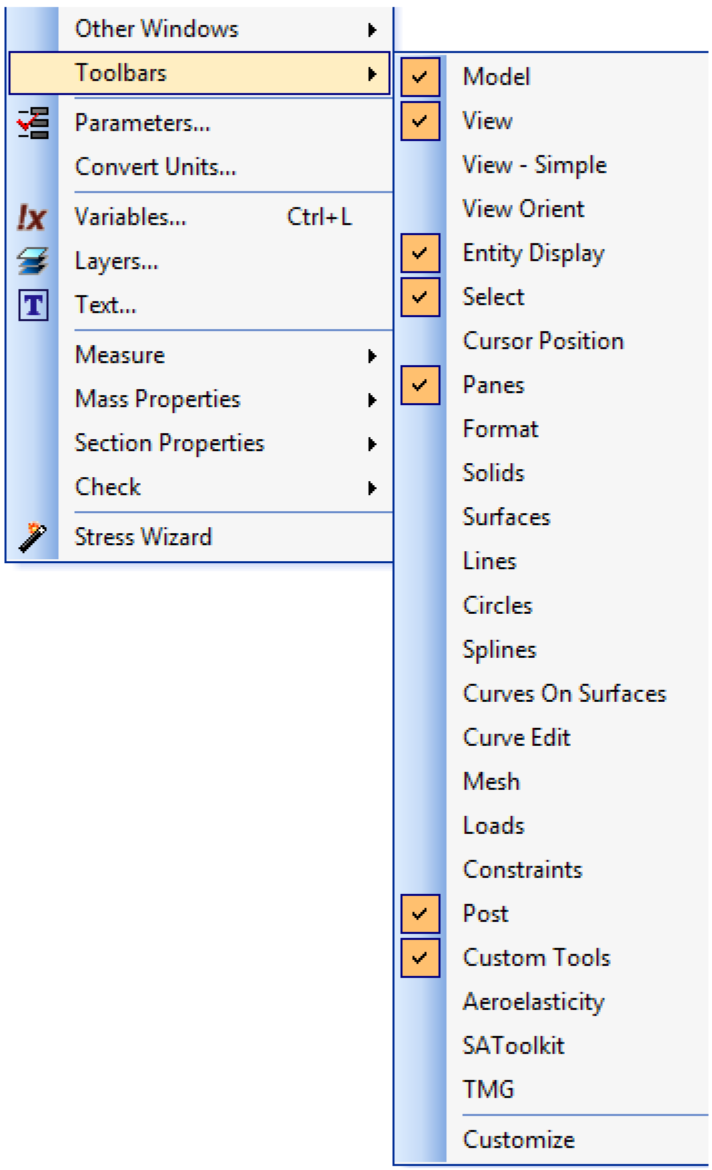
The most important function of the FEMAP main window is to provide you with easy access to the many powerful tools in FEMAP. It provides access through several methods, including the FEMAP main menu, toolbars (Standard and Custom), the dockable panes, and trays on the Status bar at the bottom right portion of the main window. Each of these areas will be discussed in more detail in Section 4.2, "Accessing FEMAP Commands".

In addition to command access, the main window also serves to identify the model currently active, as well as provide status and help information. The FEMAP title bar at the top of the window will show the name of the model which is currently active. When multiple models are open, the FEMAP title bar will display the name of the active model and the active view in the following manner, *Model Name.MOD : View Name.*



Each window in the main FEMAP graphics area has a named “Tab”. By right mouse clicking this tab, several options are available. *Create/Manage* will bring up the *View Manager* dialog box. *New* will open up a new view, while *Close* will close down the active view. *Tab Location* allows you to choose from 4 different locations for the tab (*Top*, *Right*, *Bottom*, or *Left*). T*ab Color* allows you to choose from one of eight available tab colors or choose to have no colors, while *Show Full Path* will show the full path to the directory where the active model is currently located on the tab. *Toggle Tabs* allows you to toggle the tabs on and off and *Toggle Title Bars* turns the title bars on and off, when the are visible.

The Status bar at the bottom of the FEMAP main window has several functions. When performing commands that require more than a few seconds, such as importing a large amount of analysis results, the Status bar will demonstrate the percentage completed. This provides feedback that FEMAP is still importing the file, and is still active. When not performing commands, the Status bar will serve as the menu Help location and contain trays which allow you to access FEMAP commands to activate a specific property, load set, constraint set, group, and output set.



To use menu Help, simply move your cursor to a menu or toolbar command. A brief description of the command will be provided in the Status bar location. If you maintain the cursor above one of the toolbar commands, you will also see the command name appear next to the cursor in a “Tooltip”. This is in addition to the description in the Status bar location.

You also have control over whether any number of toolbars (Standard and Custom) and dockable panes are displayed inside the main FEMAP interface.

Each toolbar can be made visible or hidden using the *Tools, Toolbars...* menu, then choosing a particular toolbar from the list. When the toolbar is visible, it will have a check mark next to the toolbar name on the menu. By default, when a toolbar is made visible for the first time, it will be placed (“docked”) at the top of the FEMAP interface and below the main menu as a starting position. Any toolbar can be moved around the edge in the FEMAP interface and remain docked in the “Toolbar Docking Areas” or be “peeled” (clicked and dragged) away from a Toolbar Docking Area to “float” inside the FEMAP interface.

Note: The Toolbar Docking Areas refer to the areas around the edge of any FEMAP Graphics windows and all open or retracted Dockable Panes in the FEMAP interface. This means you can place the toolbars above, below, to the left, or to the right of the Graphics windows and Dockable Panes.

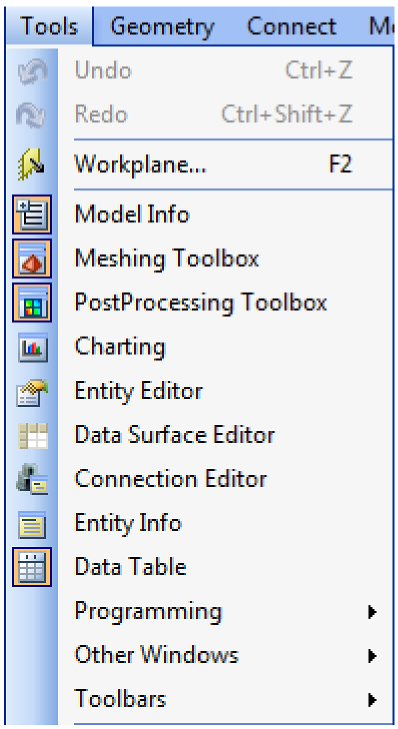
FEMAP contains “Dockable Panes” that offer different tools used to create and modify models, as well as, evaluate and sort data, create reports, and view specific entities. Each dockable pane can be either visible or hidden by selecting the specific dockable pane from the *Tools...* menu. The dockable panes are not active when hidden, so they must be made visible for use. When visible, the dockable panes can appear in one of three states: Docked, Docked with a Tab (pane is retracted until fly-out), or Floating. There are specific positions dockable panes can be docked inside the interface and these positions are marked with Docking Position Indicators which only appear when the pane is being moved around the interface. A docked pane can be retracted, in which case it will appear as a “Named Tab” around the edge of the interface or “peeled” off to float in the interface, much like the toolbars.

Both the toolbars and dockable panes are explained in much greater detail below.

The Status Bar can also be made visible or hidden using the *Tools, Status Bar* command.

4.1.2.2 FEMAP Dockable Panes

FEMAP contains several “Dockable Panes” that offer different tools used to create and modify models, evaluate and sort data, create reports, and view info of specific entities. There are others which allow you to create customized features by recording macros or creating advanced programming routines by directly accessing the FEMAP database using the FEMAP API (Applications Programming Interface). Each dockable pane can be either visible or hidden by using the *Tools...* menu command corresponding to the specific dockable pane.



The *Dockable Panes* are each individually documented in the FEMAP *Commands* manual. They are:

- **Model Info**: See Section 7.2.1, "Tools, Model Info"
- **Meshing Toolbox**: See Section 7.2.2, "Tools, Meshing Toolbox"
- **PostProcessing Toolbox**: See Section 7.2.3, "Tools, PostProcessing Toolbox"
- **Charting**: See Section 7.2.4, "Tools, Charting"
- **Entity Editor**: See Section 7.2.5, "Tools, Entity Editor"
- **Data Surface Editor**: See Section 7.2.6, "Tools, Data Surface Editor"
- **Connection Editor**: See Section 7.2.7, "Tools, Connection Editor"
- **Entity Info**: See Section 7.2.8, "Tools, Entity Info"
- **Data Table**: See Section 7.2.9, "Tools, Data Table"
- **Programming, API Programming**: See Section 7.2.10, "Tools, Programming, API Programming"
- **Programming, Program File**: See Section 7.2.11, "Tools, Programming, Program File"
- **Other Windows, Messages**: See Section 7.2.12, "Tools, Other Windows, Messages"
- **Other Windows, Status Bar**: See Section 7.2.13, "Tools, Other Windows, Status Bar"

When a dockable pane is hidden, it cannot be used until it is made visible. When visible, the dockable panes can be in one of three states: Docked, Docked with a Tab (pane is retracted until “fly-out”), or Floating.

Docked dockable panes

When a dockable pane is docked, it resides around the edge of any FEMAP graphics windows and inside of any docked toolbars or “docked and tabbed” (retracted) dockable panes. There are a number of different “docking positions” and “docking methods” in which you can “dock” a dockable pane. A dockable pane can be dragged on to any of the “docking position indicators” (arrow-like icons located within the FEMAP interface, only when a dockable pane is being moved) and once there dropped into a predetermined “docking position”. This means you can place the dockable panes above, below, to the left, or to the right of the graphics windows. You can change the size of any of the dockable panes by placing the cursor on the border of any pane until the “typical windows double-sided resizing arrow” appears, then dragging it until it reaches the size you desire.

You can “stack” the dockable panes on top of one another by dragging one pane onto the “stacked” docking position indicator inside another dockable pane. Once stacked, you can toggle between different “stacked” dockable panes by clicking on the titled tab of a specific dockable pane. When “stacked”, the dockable pane which was used last will remain visible until the tab of another “stacked” dockable pane is clicked, the active dockable pane is “peeled” off to become a floating dockable pane, or docked into a different position.

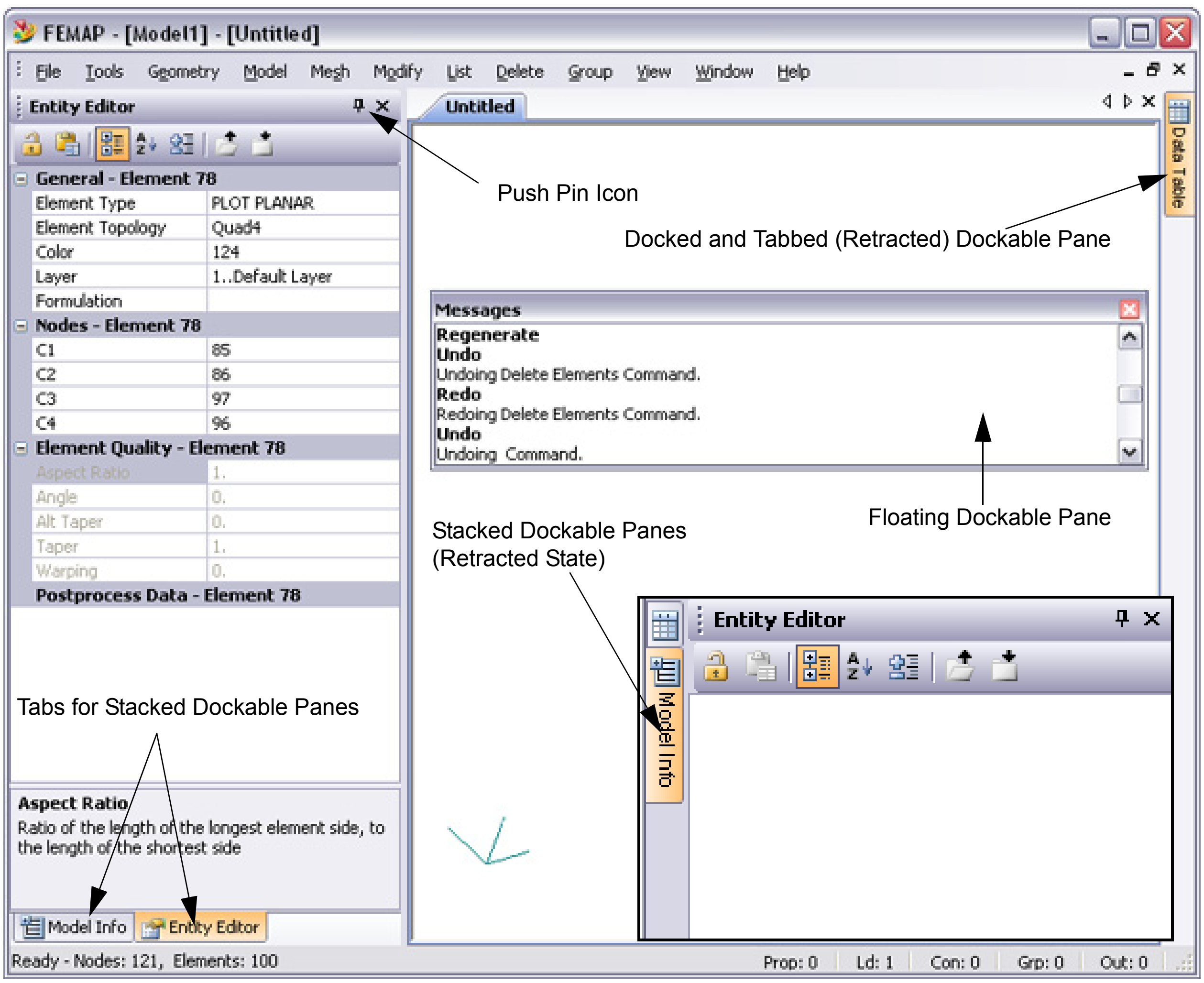
"Docked and Tabbed" (Retracted) dockable panes

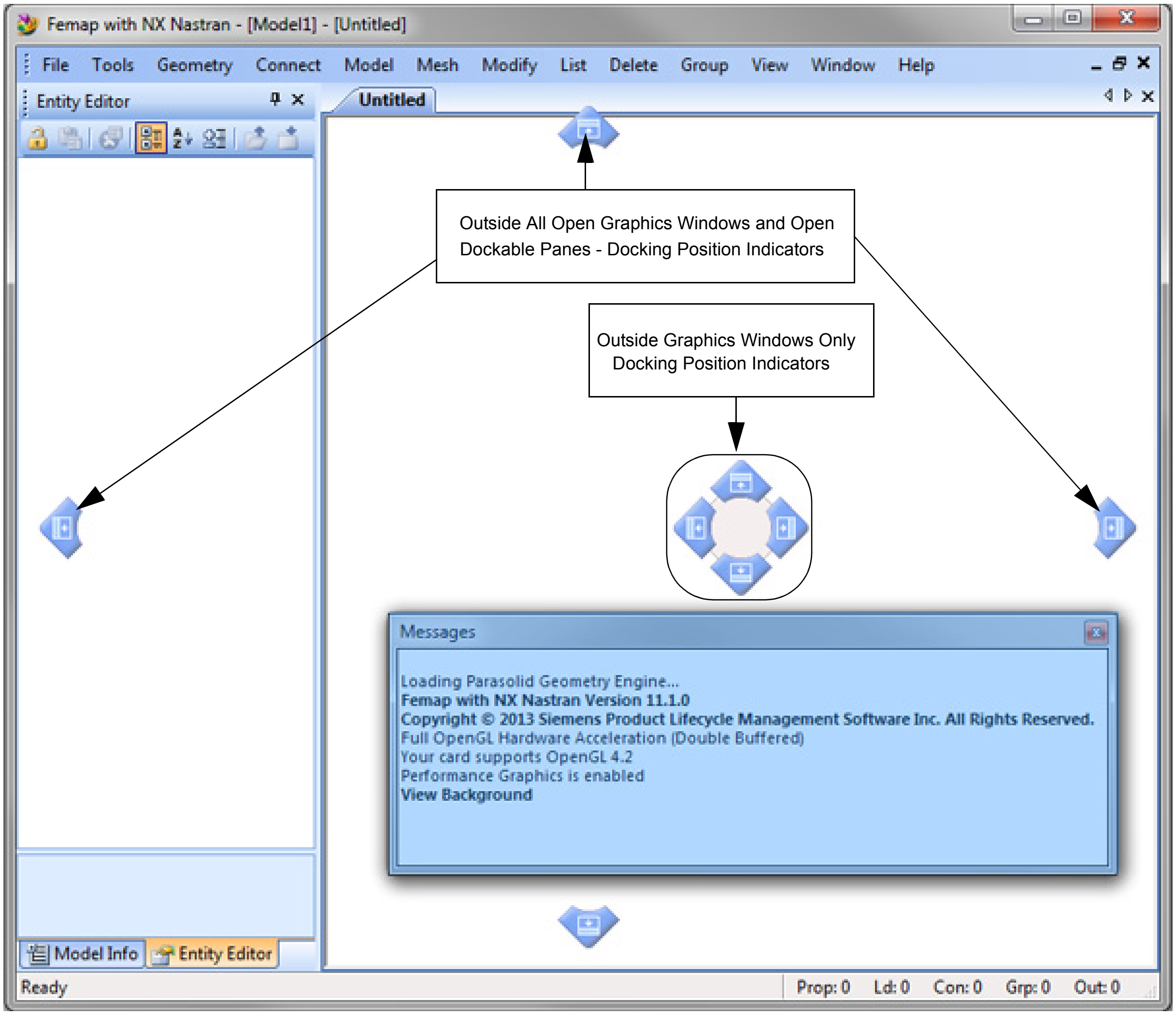
In order to have a dockable pane become "docked and tabbed", simply toggle the push pin icon on the title bar of a specific pane (the "pin" icon will appear to be "pulled out" of the screen). The pane will now appear as a titled tab on the top, bottom, left, or right (based on the pane's docked position) of all open graphics windows and other dockable panes. If you decide to "pull out" the push pin on a "stacked" set of dockable panes, they will appear as a "stacked" set of tabs, with only the most recently used pane showing the title and the other panes only showing an identifying icon.

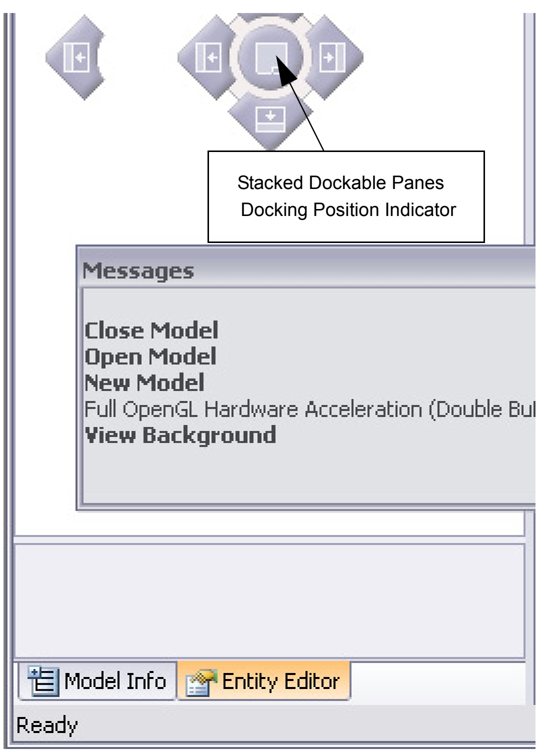
"Docked and Tabbed" (Retracted) Dockable Panes can be located on the top, bottom, left, or right of the FEMAP interface. These panes remain retracted until you place the cursor over them, at which point they will "fly out" to a certain size (fly-out size depends on which of the dockable pane is flying out). Once the dockable pane has "flown-out" it can be used, resized, and adjusted just as it would if it was simply docked or floating. You can continue to use the "flown-out" pane until you move the cursor off the pane and onto any visible graphics window, when it will "retract" back to a "tabbed" state. A "docked and tabbed" pane can become simply "docked" again by toggling the push pin icon on the title bar (the "pin" icon will now appear to be "pushed into" the screen).

Floating dockable panes

When a dockable pane is "peeled" (clicked and dragged) from a docked position and dropped anywhere on the screen other than a "docking position indicator", it becomes a "Floating Dockable Pane". A Floating dockable pane can be positioned anywhere on the desktop during an open FEMAP session. You can return a floating toolbar to a docked position by dragging it onto a docking position indicator or by double clicking the Title bar of a floating dockable pane. A floating pane can be closed by clicking the "X" in the Title bar in the upper right hand corner.







Docking Position Indicators

The docking position indicators appear when a dockable pane is being "dragged" around the FEMAP interface. The direction the arrow is pointing represents which side of the other open windows (graphics window and/or dockable panes) that a dockable pane will be docked. In order to get the window to the correct docked position, you must drag the dockable pane until the cursor is over the appropriate docking position indicator. Once the cursor is over an indicator, a preview "shadow" will appear representing the position where the dockable pane will be placed when the mouse button is let go.

There are three different sets of docking position indicators:

The "Outside All Open Graphics Windows and Open Dockable Panes" set appears every time you begin dragging a dockable pane. This allows you to dock the panes above, below, to the left, or to the right of all graphics windows and open dockable panes.

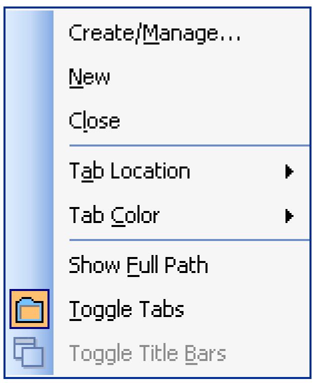
One of the other two sets of docking position indicators will also appear depending on where you have the cursor positioned inside the FEMAP interface. If you have the cursor over the graphics window, the "Outside Graphics Windows Only" docking position indicators will appear. If you have the cursor over another dockable pane, the "Stacked Dockable Panes" set of indicators will appear.

Note: A floating dockable pane can only be changed into a "docked and tabbed" pane by first docking it into position to make it a simply "docked" pane AND then toggling the push pin icon in the title bar. Vice versa, a "docked and tabbed" pane can only be changed into a floating pane by first toggling the push pin (to pushed-in status) to dock the pane, then drag it out

4.1.2.3 The FEMAP Tabbed Graphics Windows

The last type of window used by FEMAP is the Tabbed Graphics window. Just like the main window and the active Dockable Panes, one Graphics window is automatically created when you start FEMAP. By default, it will cover the area of the main window which is not occupied by the "docked" dockable panes and toolbars. Unlike the other windows, you can create multiple graphics windows (*Window, New Window* command) if you want to see multiple views of one or multiple models in FEMAP.

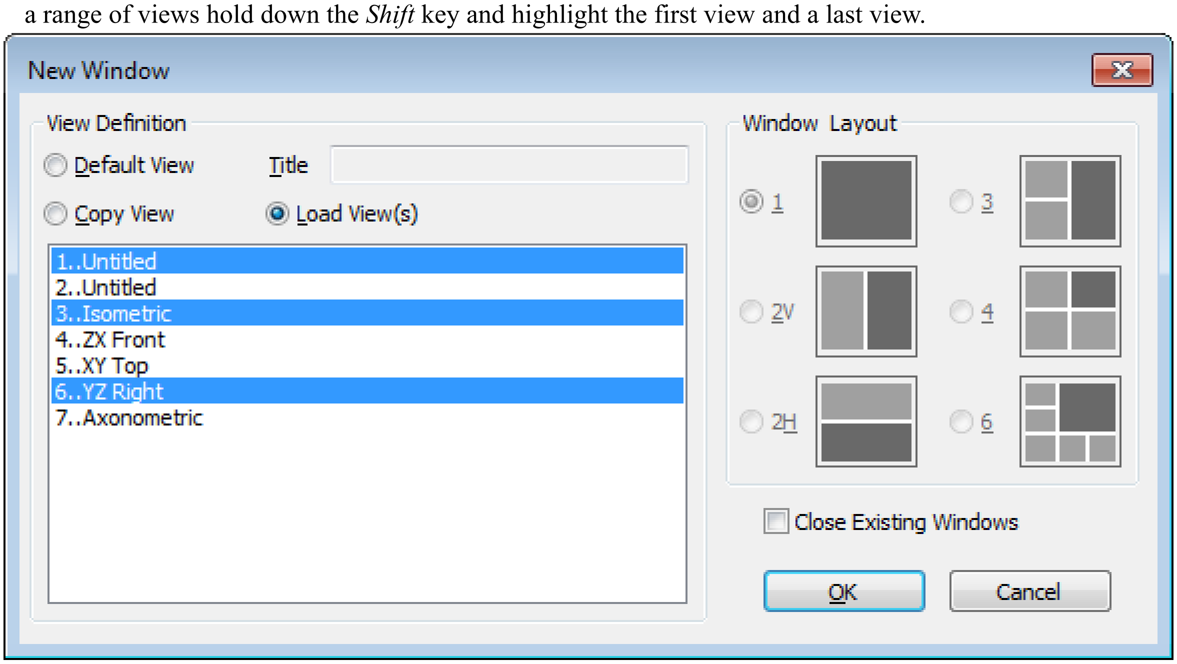
At the top of each view, there will be a solid color Tab which will show the name of that view. When multiple models are open, the tab will give the Model name and view name in the following format: *Model Name.MOD : View Name.* If you would like to turn the tabs off and on you can use the *Window, Toggle Tabs* command or right mouse click on the tab and select *Toggle Tabs*.



When you right mouse click on a Tab you will also find options to set the active view, create a new view, close the view, change the tab location (*Top*, *Right*, *Bottom*, or *Left*) or color, show the "Full Path" to the directory where the model is currently being stored, and toggle the title bar (when title bar is visible).

The FEMAP *View* and *Window* menu commands control all aspects of graphics windows. There is a one-to-one correspondence between the FEMAP views and the graphics windows. Every graphics window has its own unique FEMAP view, which is stored with your model. When a graphics window is originally created using the *Window, New Window* command, a FEMAP view is created in the model database. When that window is closed, the view remains in your database (unless you use the *Delete, View* command). To open one window at a time you simply use the *View, Create/Manage* command and select the desired view. The window will reappear in its former size and position, using the same options as when it was last displayed, while closing the view that was previously active.

If you would like to turn on many closed views at the same time, use the *Window, New Window* command. Choose *Load Views* in the *View Definition* section of the New Window dialog box and then select the desired views from the list of current views. To select multiple views one at a time, hold down the *Ctrl* key while selecting or to select a range of views hold down the *Shift* key and highlight the first view and a last view.



Just like the *Messages* window, the contents of graphics windows can be exported to a file, or to other Windows applications. For graphics windows, however, you must use the *File, Picture, Save...* or *File, Picture, Copy...* commands.

Sizing the Graphics Windows

Also like the dockable panes, there are several ways to size the graphics windows including grabbing the border, pressing maximize, the *Window, Tile Horizontal*, *Window, Tile Vertical*, or *Window, Cascade* commands.

4.2 Accessing FEMAP Commands

As mentioned in the FEMAP main window description, there are several ways of accessing FEMAP commands. Most of these are contained on the main window, such as the main menu, toolbars, and Status bar. There are several other methods as well, including using the right mouse button, as well as keyboard input. There are eight basic methods of accessing FEMAP commands.

- Main menu - See Section 4.2.1, "FEMAP Main Menu"
- Toolbars (Standard and Custom) - See Section 4.2.2, "FEMAP Toolbars"
- Quick Access menu (right-mouse button in graphics window when Select Toolbar is not active) - See Section 4.2.3, "Quick Access Menu (Right Mouse Button)"
- Shortcut keys - See Section 4.2.4, "Shortcut Keys"
- Status bar - Section 4.2.5, "Status Bar"
- The Select Toolbar (alternative gateway to many useful commands) - See Section 4.2.6, "The Select Toolbar"
- Context Sensitive menus - Section 4.2.7, "Context Sensitive Menus"
- Dockable Pane icons - Section 4.2.8, "Dockable Pane Icons"

4.2.1 FEMAP Main Menu

At the top of the main window, under the Title bar, is the FEMAP menu. This menu provides access to all of the available commands. You can execute these commands through any of the standard Windows methods - picking with the cursor/mouse, pressing *Alt* and one of the underlined letters or the direction keys, or by using one of the shortcut/accelerator keys shown to the right side of the menu.

Often, some commands on the menu are displayed in gray. These commands are temporarily disabled. For example, the *List, Nodes* command is disabled if you do not have any nodes in your model to list. Disabled commands will automatically enable themselves when the data they need is available in your model.

All of the commands shown on the menu bar (at the top of the main window) cause another menu to “drop-down” to display additional commands. You will notice that some of the commands on this drop-down menu have a small arrow on the right side of the menu. Selecting one of these commands will display a third menu level. The FEMAP main menu never goes below this third level, and many commands are at the second level. This minimizes the time and effort involved in selecting commands. Each command is documented in the *FEMAP Commands*. This manual will concentrate more on the general use of FEMAP.

4.2.2 FEMAP Toolbars

FEMAP has many useful toolbars that help you perform a variety of different functions. The toolbars contain icons representing certain commands and are grouped together by functionality. Each toolbar can be made visible or hidden using the *Tools, Toolbars...* command, then choosing a particular toolbar from the list. When the toolbar is visible, it will have a check mark next to the toolbar name on the menu. By default, each toolbar will be placed (“docked”) at the top of the FEMAP interface and below the main menu as a starting position.

Once the toolbars are visible, they can either be “docked” around the edge of the FEMAP interface or “floating” somewhere inside the FEMAP interface. By default, when a toolbar is made visible for the first time, it will be docked, but is can be “peeled” (clicked and dragged) away from the edge of the FEMAP interface and become a floating toolbar. Aside from having a docked or floating toolbar, there are several other options, which are explained here.

4.2.2.1 Toolbar Types

Docked toolbars

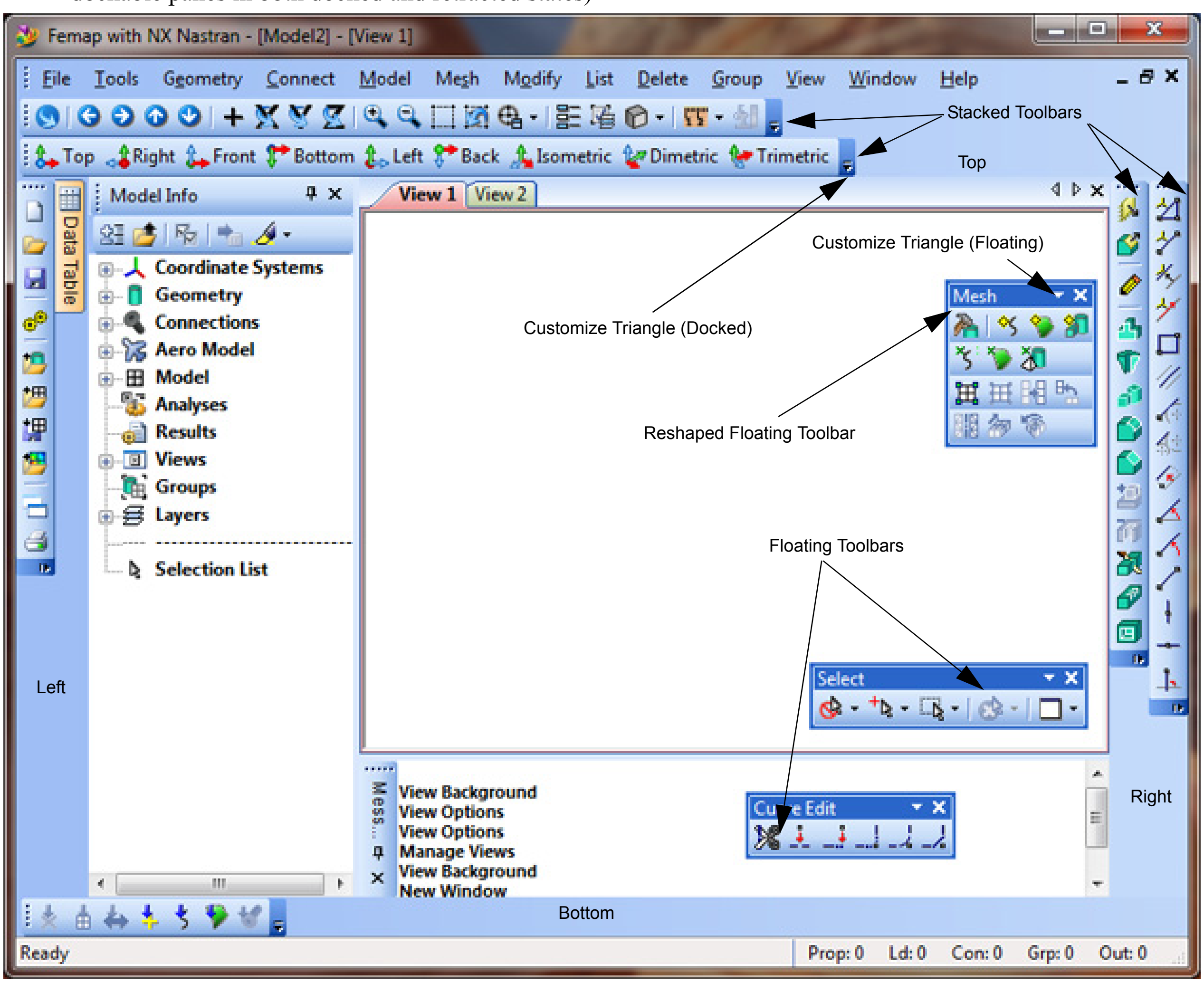
When a toolbar is docked, it resides around the edge of any FEMAP graphics windows and all open or retracted "Dockable Panes" (Entity Editor, Model Info, Data Table, or Messages windows) of the FEMAP interface. This area is referred to as the "Toolbar Docking Area". A toolbar can be dragged anywhere inside the Toolbar Docking Area and still remain docked. This means you can place the toolbars above, below, to the left, or to the right of the graphics windows and Dockable Panes. Also, any toolbar can be "stacked" above or below horizontal toolbars or to the left or right of vertical toolbars. Docked toolbars can be shifted around as well to create a user-defined configuration.

Floating toolbars

When a toolbar is "peeled" (clicked and dragged) off the edge of the FEMAP interface and placed on top of the graphics window or dockable panes, it is now a floating toolbar. A floating toolbar can be positioned anywhere you would like to put it within the limits of an open FEMAP session. You can return a floating toolbar to a docked position by dragging it back onto the edge of the FEMAP interface or by double clicking the Title bar of a floating toolbar. A floating toolbar can be closed by clicking the "X" in the title bar in the upper right hand corner. If reopened, the floating toolbar will appear in the last position it was in before being closed.

A floating toolbar can be also "reshaped" to better fit your modeling needs. To reshape a floating toolbar, place the cursor over the edge of a toolbar (you will see a two-headed resizing arrow common to many windows programs), click the mouse, and drag the toolbar into the desired rectangular shape. When a "reshaped" floating toolbar is docked, it will return to the original shape while docked. If it is later undocked (or reopened), it will appear in the "reshaped" configuration.

The following figure shows some sample positions you can place docked and floating toolbars. (It also shows dockable panes in both docked and retracted states)



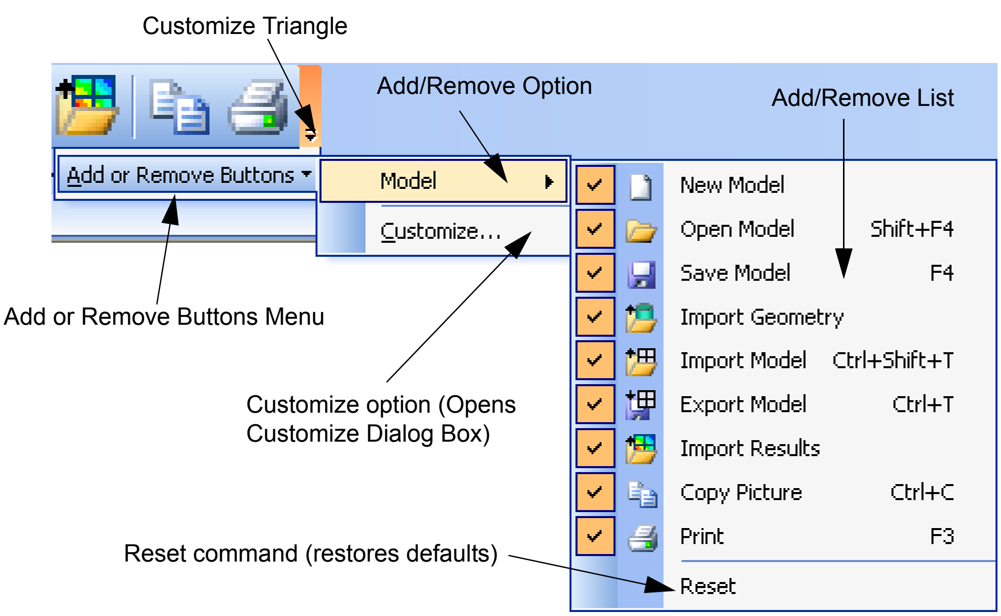
Note: The toolbars can be turned on and off more than one place. The most obvious way to turn them off is through the *Tools, Toolbars, ...* command. A second way to turn them on and off is by clicking the right mouse button anywhere in the "Toolbar Docking Area" around the edge of the FEMAP interface, which will bring up a menu of toolbars to switch on and off one at a time. The third way to turn the toolbars on and off is by clicking the right mouse button on the title bar of a floating toolbar. Finally, the last place to turn them on and off is when using the "Customize" menu available on all the toolbars and described in greater detail in the "Customizing toolbars" section.

4.2.2.2 Customizing toolbars

FEMAP gives you the ability to customize the toolbars in several different ways.

The simplest form of customization available on the toolbars is repositioning the currently visible icons on the toolbars which are currently open. In order to reposition an icon to a new position, hold the "Alt" key down, left mouse click the icon you would like to move, then drag it to a new position on either the icon's original toolbar or any other open toolbar. Let the mouse button go and the icon will be dropped into the new position. You can also use this process to remove icons from a toolbar by dragging a chosen icon to a place on the screen with no toolbars and then dropping it there. A small "x" will appear next to "dragged" icon when it is in a position where it can be dropped and removed.

All other customization begins by clicking on the small triangle ("Customize" triangle) that is on every visible toolbar (the triangle appears in a different place depending on whether the toolbar is docked or floating). When the "Customize" triangle is clicked, a menu will drop down which says "Add or Remove Buttons". When the "Add or Remove Buttons" menu is highlighted, it will bring up a second level menu with two options; Add/Remove from any of the toolbars currently in the same toolbar "row" or "Customize".



Add/Remove option

The add/remove option will show the name of the toolbars currently in the same row, which when highlighted will bring up another menu level which allows you to individually turn existing icons on or off (You can turn multiple icons on or off while the menu is open and the toolbar will dynamically change). When the icon and command name have a check mark next to them, the icon is visible on the toolbar. To restore the default settings for a toolbar, choose Reset at the bottom of the menu.

Customize... option

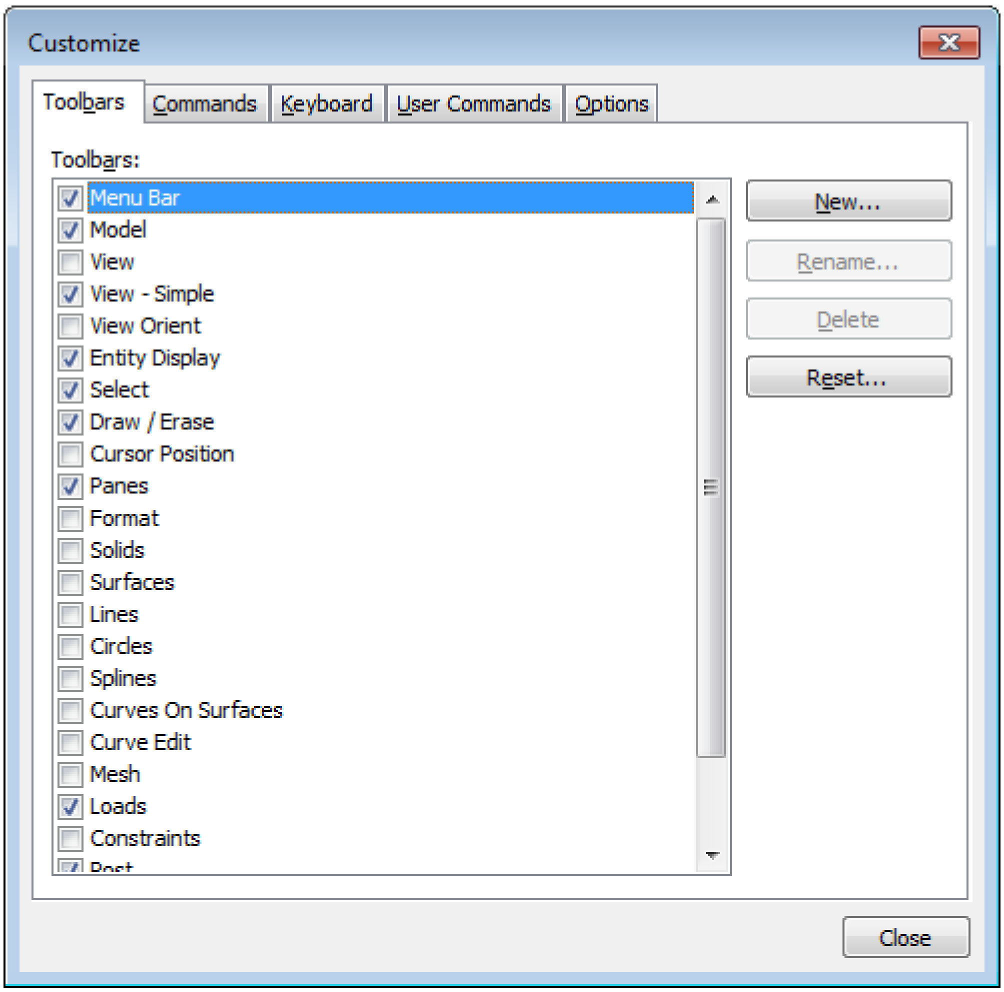
The *Customize...* option will bring up the *Customize* dialog box when clicked. Once open, this dialog box contains five different tabs which represent various methods to customize your toolbars. Also, while the *Customize* dialog box is open, you can right mouse click on any icon in any visible toolbar and a “Customize Icon” menu will appear. We will discuss the Customize dialog box and Customize Icon menu in greater detail below.

Customize Dialog Box

...The *Customize* dialog box is broken into five different sections: *Toolbars*, *Commands*, *Keyboard*, *User Commands*, and *Options*. Each of these sections pertains to a specific area of toolbar customization. There is a tab for each heading that can be clicked to bring up the specific options for each section.

Toolbars

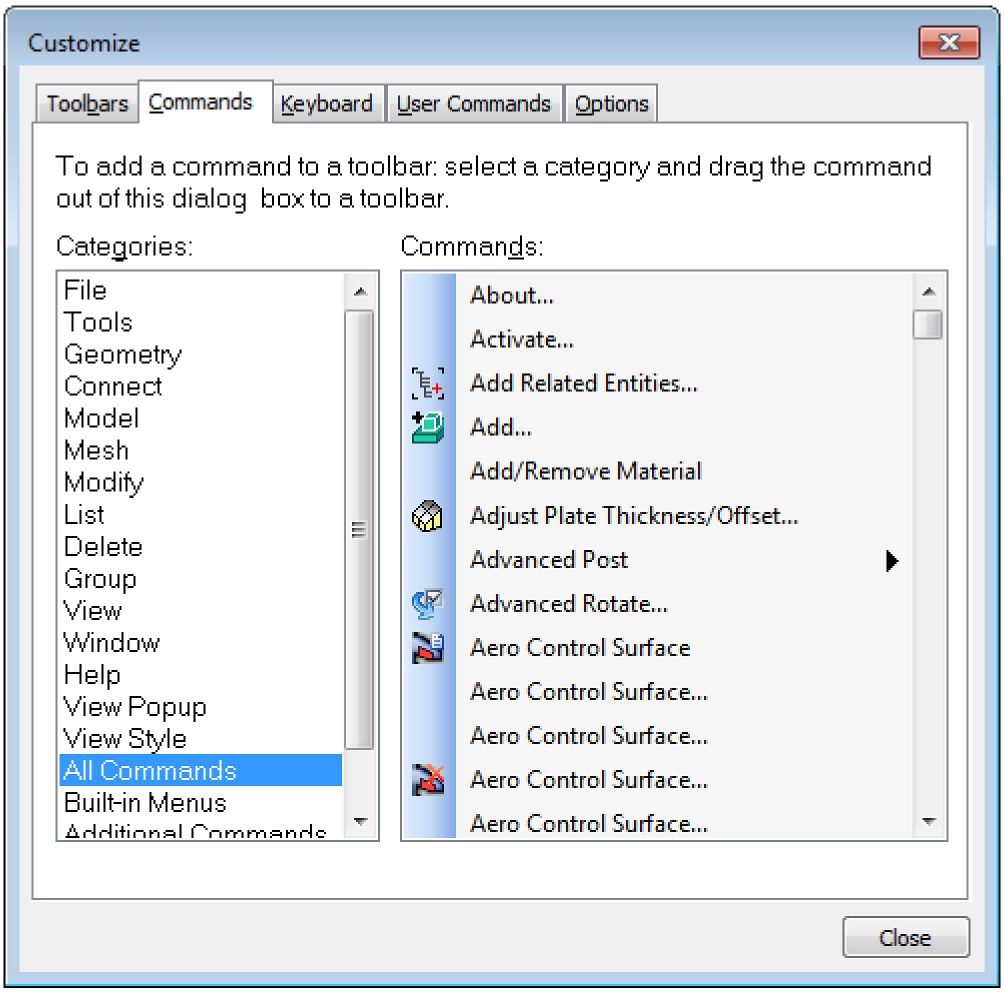
...Allows you to turn toolbars on and off by clicking the check box next to the toolbar name. This allows you to turn multiple toolbars on and off while in the same command.



As each toolbar is checked or unchecked, it will appear or disappear in the FEMAP interface. This tab also allows you to create new, personalized toolbars by pressing the *New* button. FEMAP will prompt you to give the new toolbar a name and will bring up a “blank” toolbar in the FEMAP interface. You can then add icons for exiting commands or user commands to the new toolbar. “Personalized” toolbars can be renamed at any time using the *Rename* button or deleted using the *Delete* button. Using the *Reset* button will reset the toolbar currently highlighted in the list to the default configuration.

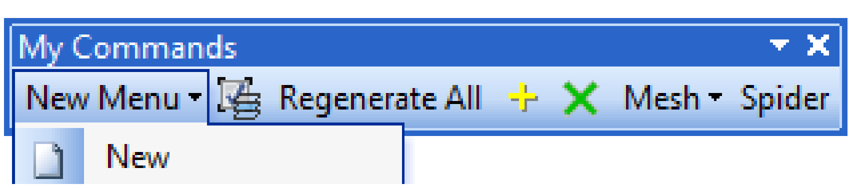
Commands

...The *Commands* tab contains all the commands available in FEMAP through the Main Menu structure. Choose the type of command you are looking for from the *Categories* list, then locate the specific command in the *Commands* list. Once the specific command is located, click and hold the left mouse button to "grab" the command. Now you can drag the "grabbed" command onto a visible toolbar and place it on that toolbar. Along with the commands available through the Main Menu structure, categories such as "Additional Commands" and "View Popup" allow access to specific view options and "right mouse menu" selections. You may also add an entire existing FEMAP menu to a toolbar using the "Built-in Menus" category or create a new menu of existing and user commands by dragging the New Menu command onto a toolbar and then filling the blank menu with commands. Any user commands will show up in the "User Commands" category. Any combination of icons and commands can be put together on a "personalized" toolbar.



Many commands have icons which do not appear on any existing standard toolbar. These icons are in FEMAP specifically so you can add commands to existing toolbars and create your own "personalized" toolbars.

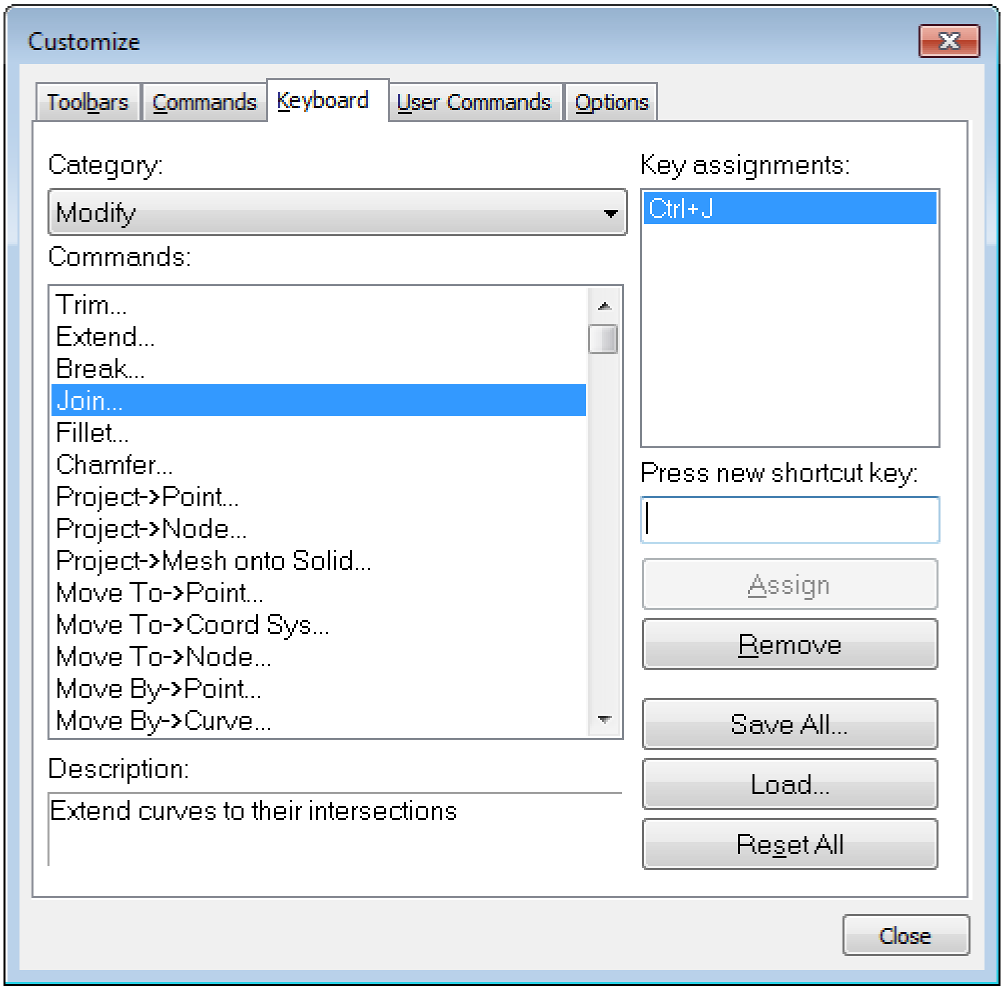
An example of a "personalized" toolbar can be seen in the next figure. Notice that there is a "New Menu" containing a few existing commands from different menus and toolbars that appear on a drop-down menu. Also included on this Custom toolbar are the *Visibility* icon from the "View" category, *View Regenerate All* command from the "Additional Commands" category, the *Snap to Point* and *Snap to Node* icons from the "View Popup" category, the entire *Mesh* menu from the "Built-in Menus" category, and *Spider* (a user command) from the "User Commands" category.



Keyboard

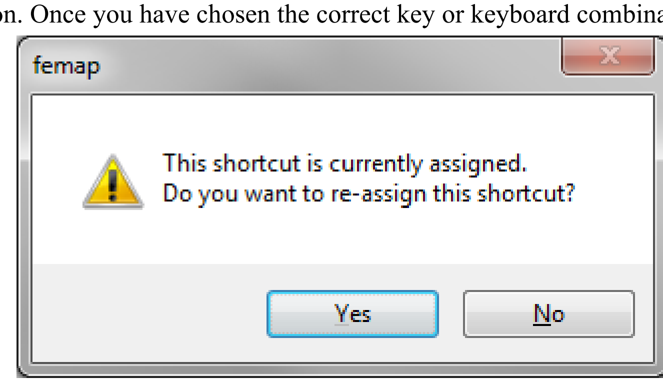
This option allows you to define letter keys in FEMAP as FEMAP commands. You can also assign currently unused function keys and keyboard combinations (i.e., CTRL, SHIFT, ALT + letter or function keys) as FEMAP commands as well.

You can therefore quickly customize FEMAP to use letter and function keystrokes, as well as keyboard combinations, as your most often used FEMAP commands.



This option allows you to define any of the keys on your keyboard and keyboard combinations as FEMAP commands, thereby enabling you to define many different shortcut keys.

To define a shortcut key, first choose the *Category* from the drop down list, then highlight the command from the *Commands* list. After the command is highlighted, click in the "Press new shortcut key:" field and press a key or keyboard combination. Once you have chosen the correct key or keyboard combination, click the *Assign* button.



If the key or keyboard combination has already been defined, FEMAP will let you know and bring up a dialog box stating "This shortcut is currently assigned. Do you want to re-assign this shortcut?" By clicking the *Yes* button, the key or keyboard combination will be added to the "Key assignments:" list and REMOVED from the command that was previously using that shortcut key or keyboard combination. Clicking the *No* button allows you to select an unused shortcut key or keyboard combination and leaves all other shortcut keys unchanged.

Shortcut keys can be saved by clicking the *Save All* button. FEMAP will prompt you to create a "Keyboard Shortcut File" (*.KEY file). This file will contain all of the keyboard shortcuts you have currently set in FEMAP. You can then click the *Load* button to load a *.KEY file and your shortcuts will be restored. For FEMAP versions 9.3 and above, you can load a *.KEY file from the previous version and quickly customize the new version.

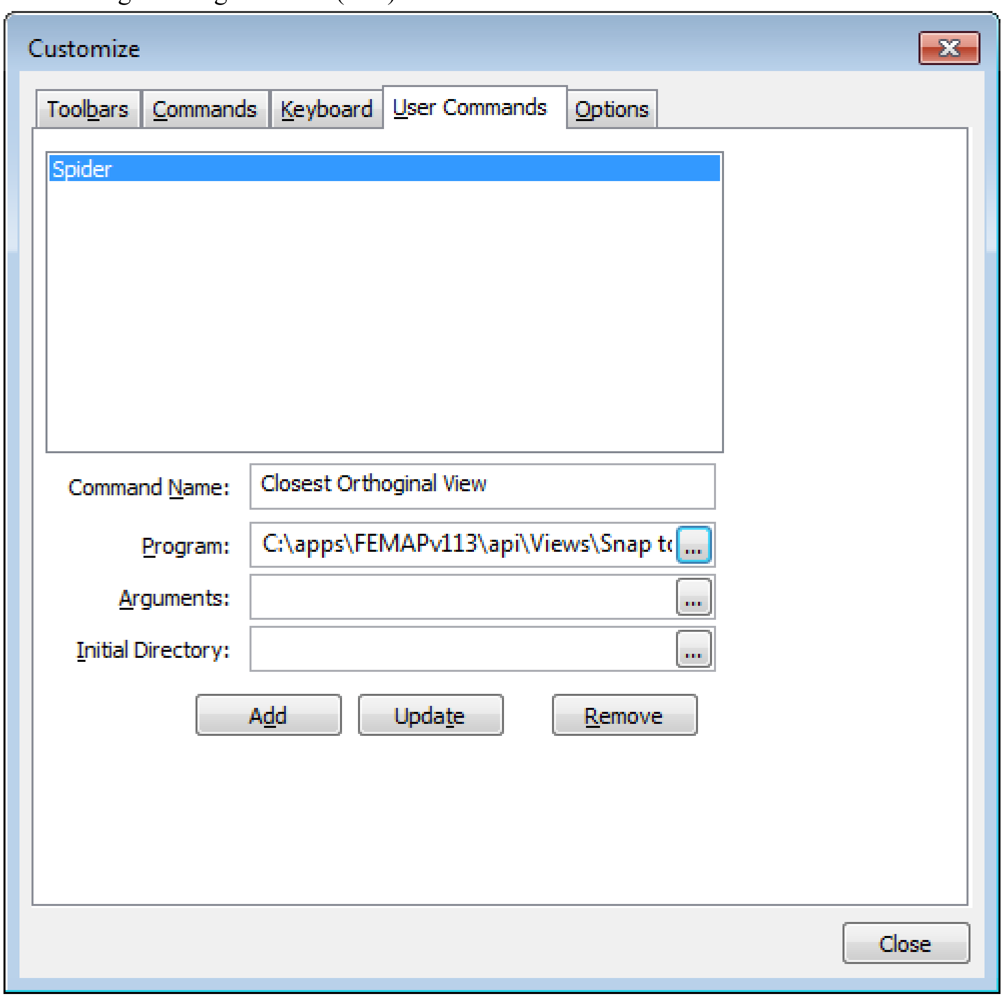
Shortcut keys can be manually removed by highlighting a key or keyboard combination from the "Key Assignments:" list and then clicking the *Remove* button. The *Reset All* button will return all shortcut keys to their default commands.

Defining shortcut keys for your most used commands, you can save time moving through the FEMAP menu structure. Shortcut keys are only available from the FEMAP menu level. If you are already in another command or dialog box, pressing these keys will not have the desired effect. In most cases, it will simply result in typing the letter which was pressed.

Hint: If you are typing in the *Messages* window, anytime you type a shortcut key, the command will be invoked.

User Commands

...The *User Commands* tab allows you to create command names for user commands created using the FEMAP Applications Programming Interface (API).



In order to locate a file to be used as a program, you can browse through windows directories using the "..." browse button next to the *Program* field. Choose the file to be used as the "program" file, click OK, and then the entire directory path will be shown in the *Program* field. There are several different files which can be used as a "Program" files including Executable (*.exe), Command (*.com), Information (*.pif), and Batch (*.bat, *.cmd) files

Once the file for the actual command has been located, the command must be given a unique Command Name. After the command has been given a name, click the *Add* button to place it into the list of User Commands.

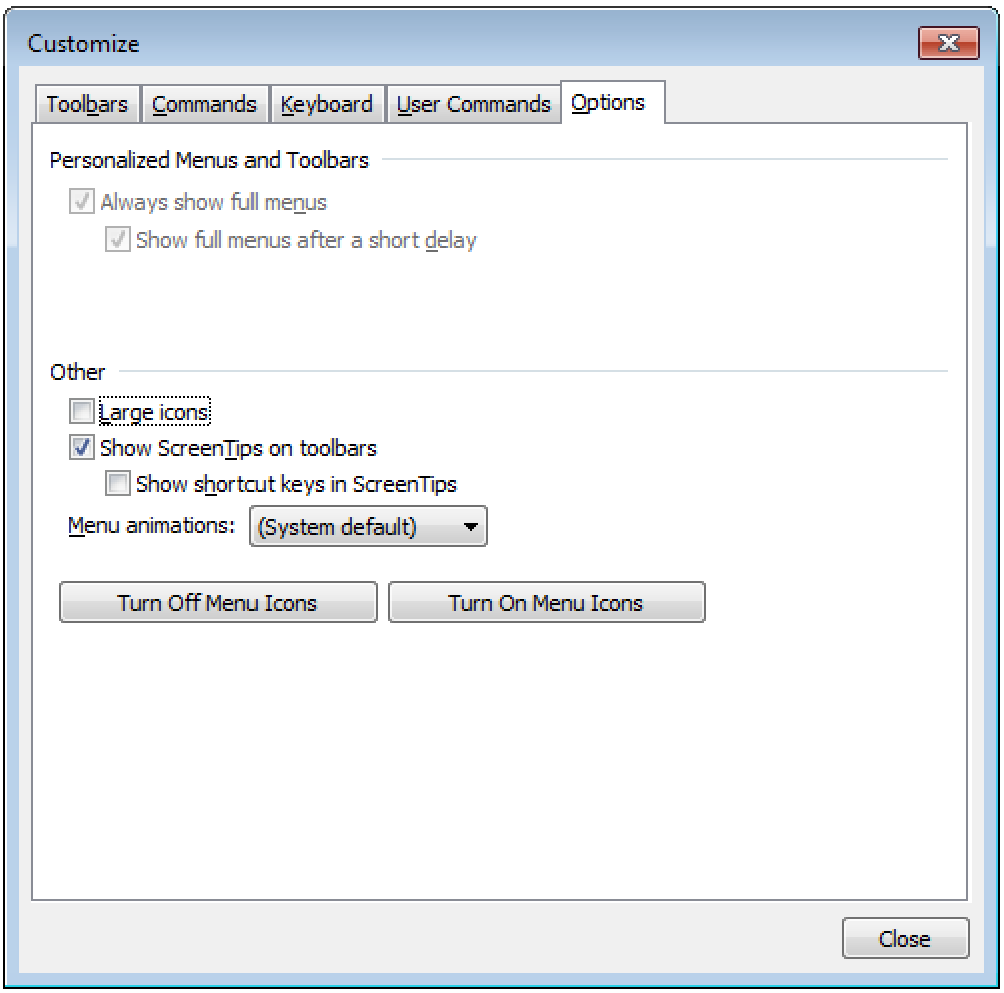
If you would like to change the name or directory path of a User Command, highlight it in the list, make any modifications, then click the *Update* button to confirm the change. To remove a User Command from the list, highlight it, then click the *Remove* button.

Along with the "Program" file itself, you may optionally enter other necessary files and command line entries into the *Arguments* field. In addition, if any program file needs to use an external directory, the path to that directory can be entered into the *Initial Directory* field.

Once the commands and are added to the User commands list, they will appear in the "User Commands" category in both the *Commands* and *Keyboard* sections of the Customize dialog box. User commands can now be added to existing toolbars or "Personalized" toolbars using the methods described in the *Toolbars* and *Commands* sections.

Options

....Allows you to select options to make the toolbars more useful. At the current time, the "Personalized Menus and Toolbars" options in the Options tab have no effect on any existing or custom FEMAP menus or toolbars. These options will be available in future versions.



To make the icons on all the toolbars larger, select the "Large icons" option.

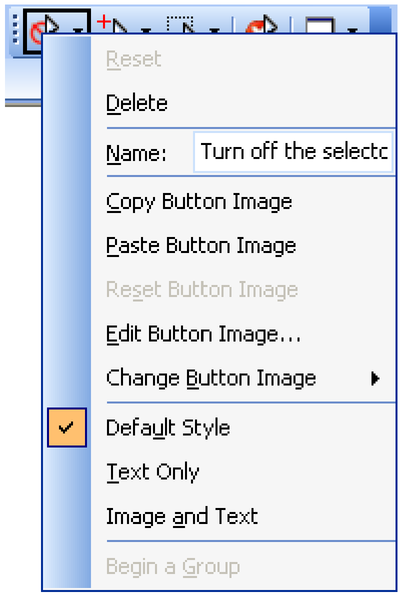
By default, the "Show ScreenTips on toolbars" option is on, you can uncheck the box to turn the ScreenTips off. If you would like the ScreenTips to also show all associated shortcut keys, use the "Show shortcut keys in ScreenTips" option.

You can select the style of how the menus drop-down by selecting a style from the drop-down "Menu animations" list. The options are (System default), Unfold, Slide, Fade, or None for a particular style or choose Random, for a different "drop down" style each time.

You can turn off all of the icons in the menus using the *Turn Off Menu Icons* button.

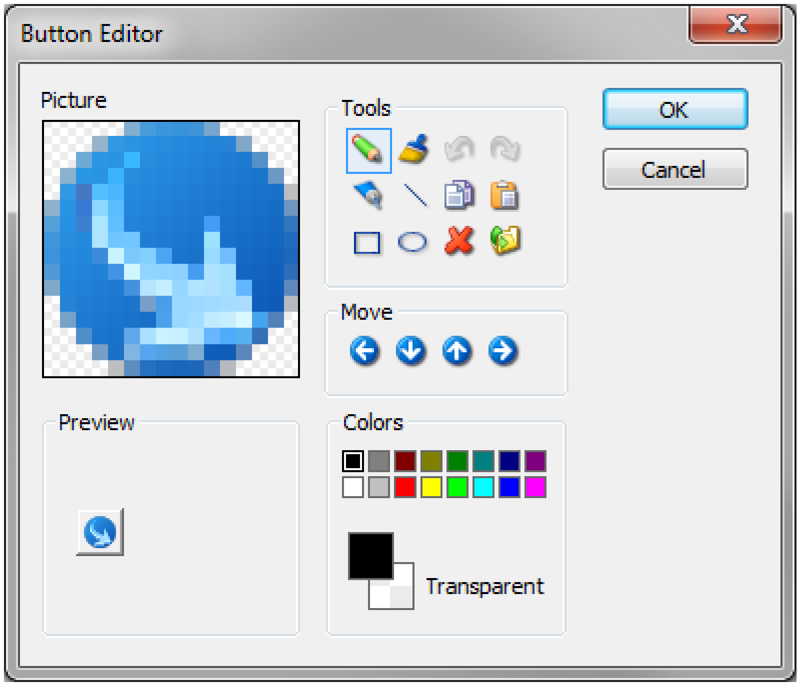
Customize Icon menu

...The Customize Icon Menu is available only when the *Customize* dialog box is open. In order to use the commands on the Customize Icon Menu, right mouse click any icon on any visible toolbar. Only the icon that you selected will be altered by the commands on the Customize Icon Menu. This menu contains commands used to delete icons from a toolbar, reset the default icon, and change the name of an icon. It also allows you to copy, paste, reset, edit, or change the button image of an icon. Along with these functions, icon style can be selected, and icons can be separated into “groups” on toolbars using partitions.



A brief description of the commands on the Customize Icon Menu:

- **Reset:** Resets all icon options (name, button image, style, group) to default values.
- **Delete:** Removes icon from the toolbar it is currently on. If the icon appears on multiple toolbars, it will only be deleted from the toolbar that you initially right mouse clicked to open the Customize Icon Menu.
- **Name:** Allows you to change the name of an icon. This name will appear on the toolbar when the Icon style is set to Text Only or Image and Text
- **Copy Button Image:** Copies the button image to the clipboard.
- **Paste Button Image:** Once an icon image is on the clipboard, it can be pasted onto to another icon to replace that icon’s current image.
- **Reset Button Image:** Resets the button image to the default button image.
- **Edit Button Image:** Brings up the *Button Editor* dialog box. In this dialog box, there are many tools to alter the appearance of a button image. Button Image size is limited to a 16 X 16 square “picture”. The existing picture can be modified by changing the colors or moving the image, a new picture can be drawn, a copied button image can be pasted in, or a picture from a file can be imported. Any combination of these methods can be used to create custom icons. There is a preview window that dynamically changes as you modify the icon and Undo and Redo tools to help modification. Once the image is finished, it can be copied to the clipboard as well.



Any imported image will be reduced to a 16 X 16 pixel resolution image, so be sure to inspect all imported images to make sure they still resemble the image after the resolution reduction.

- **Change Button Image:** Allows you to choose a button image from a set of 110 images provided by FEMAP.
- **Default Style:** Resets the icon style to the default setting. (Usually Button Image only)
- **Text Only:** Shows Icon Name only (no Button Image)
- **Image and Text:** Shows both the Button Image and the Icon Name together. (View Orient toolbar default)
- **Begin a Group:** When checked, creates toolbar partition line to the left (horizontal toolbars) or above (vertical toolbars) the icon being customized.

Standard toolbars

There are 23 “standard” toolbars that can be made visible from the *Tools, Toolbars...* command. The Standard Toolbars are listed below. For more information, see the referenced section of the *FEMAP Commands Manual*

Model Toolbar - Section 7.3.1.1, "Tools, Toolbars, Model"

View Toolbar - Section 7.3.1.2, "Tools, Toolbars, View"

View - Simple Toolbar - Section 7.3.1.3, "Tools, Toolbars, View - Simple"

View Orient Toolbar - Section 7.3.1.4, "Tools, Toolbars, View Orient"

Entity Display Toolbar - Section 7.3.1.5, "Tools, Toolbars, Entity Display"

Select Toolbar - Section 7.3.1.6, "Tools, Toolbars, Select"

Draw/Erase - Section 7.3.1.7, "Tools, Toolbars, Draw/Erase"

Cursor Position Toolbar - Section 7.3.1.8, "Tools, Toolbars, Cursor Position"

Panes Toolbar - Section 7.3.1.9, "Tools, Toolbars, Panes"

Format Toolbar - Section 7.3.1.10, "Tools, Toolbars, Format"

Solids Toolbar - Section 7.3.1.11, "Tools, Toolbars, Solids"

Surfaces Toolbar - Section 7.3.1.12, "Tools, Toolbars, Surfaces"

Lines Toolbar - Section 7.3.1.13, "Tools, Toolbars, Lines"

Circles Toolbar - Section 7.3.1.14, "Tools, Toolbars, Circles"

Splines Toolbar - Section 7.3.1.15, "Tools, Toolbars, Splines"

Curves On Surfaces Toolbar - Section 7.3.1.16, "Tools, Toolbars, Curves On Surfaces"

Curve Edit Toolbar - Section 7.3.1.17, "Tools, Toolbars, Curve Edit"

Mesh Toolbar - Section 7.3.1.18, "Tools, Toolbars, Mesh"

Loads Toolbar - Section 7.3.1.19, "Tools, Toolbars, Loads"

Constraints Toolbar - Section 7.3.1.20, "Tools, Toolbars, Constraints"

Post Toolbar - Section 7.3.1.21, "Tools, Toolbars, Post"

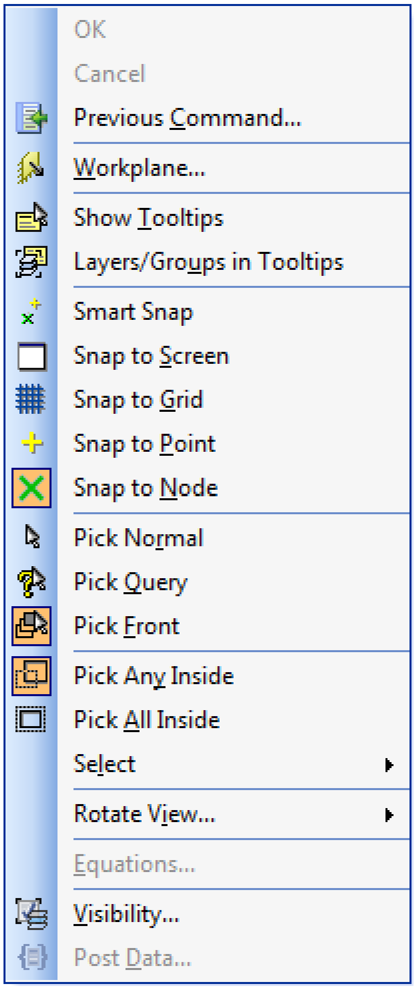
Custom and User Tools Toolbar - Section 7.3.1.22, "Tools, Toolbars, Custom and User Tools"

Aeroelasticity Toolbar - Section 7.3.1.23, "Tools, Toolbars, Aeroelasticity"

Some toolbar commands can be accessed at any time - even while you are in the middle of another command. Of special note are all of the commands on the *View Toolba*r (*Dynamic Rotate, Pan, Zoom, Model Style, View Select, View Style, etc.)* and the “Snap Modes” on the *Select Toolbar*. These commands allow you to dynamically orient your model in the active view with just a few mouse clicks. These commands are very powerful for positioning your model while in other commands and are most useful for graphical selection of your entities. Since they can be accessed while in other commands, you can actually change orientations in the middle of the selection process to obtain a better angle for picking the appropriate entities. Utilizing the *Dynamic Rotate* and other *View Toolbar* commands can significantly reduce the time required to graphically select entities.

Note: *View Toolbar* commands are available at any time during FEMAP, even in the middle of another command. The only exception is that no *View Toolbar* commands are available if you are in any other View command.

4.2.3 Quick Access Menu (Right Mouse Button)



The right mouse button provides another option to access certain FEMAP commands that are used often. Just like the toolbars, these commands can be accessed while in other commands, or as their own command. Simply point the cursor inside (not in the title bar or border) any graphics window, or inside the FEMAP main window, and press the right mouse button. A small menu will appear on your screen at the cursor location. You can choose any of the shortcut commands from this menu with either the keyboard or the left mouse button.

This shortcut menu will not appear when an entity type is active in the *Select Toolbar*. Instead, a context sensitive menu will appear giving you quick access to frequently used commands related to the active entity type. If you would like to override the context sensitive menus to display the Quick Access Menu while in the graphics window, hold down Alt, then click the Right Mouse Button. (For more information about the context sensitive menus available with the *Select Toolbar*, see Section 7.3.1.6, "Tools, Toolbars, Select")

This shortcut menu also cannot be accessed if you have the cursor in any of the dockable panes. In fact, most of the dockable panes have special context sensitive menus which will appear and perform functions specific to the dockable pane your cursor is currently inside. (For more information about the context sensitive menus available with the in the dockable panes, see Section 7.2.12, "Tools, Other Windows, Messages", Section 7.2.9, "Tools, Data Table", and Section 7.2.1, "Tools, Model Info")

Note: Some commands below will be "grayed out" on the Quick Access menu at certain times. If a command is not currently available, it will be grayed out and non-selectable. For instance, post data will be grayed if you are in another command, and *OK* and *Cancel* will only be selectable when in a command.

The commands that are on the shortcut menu are described in the following table

Command	Description
OK	only available while you are in a command dialog box. It simply presses the dialog box *OK* button.
Cancel	only available while you are in a command dialog box. It simply presses the dialog box *Cancel* button.
Previous	only available when not in another command. It accesses the last menu command.
Stop API Tool	only available when an API script is currently running. Stops the API script and displays information about what might happen to the model when stopping the API script.
Workplane...	same as the *Tools, Workplane* command. It lets you redefine the location and orientation of the workplane
Show Tooltips	when this mode is activated, a Tooltip note will pop up with useful information about the entity which is currently highlighted. To toggle this option off, select it from the menu again. It can also be turned on and off using the *Selector Modes* menu of the *Select Toolbar*. (see Section 7.3.1.6, "Tools, Toolbars, Select" and Section 5.11.3.1, "Show Tooltips" of the *FEMAP User Guide*)
Layers/Groups in Tooltips	when on, includes Layer and Group information in the Tooltip note.
Smart Snap	snap to the nearest node, point, midpoint of a curve, or center point of an arc based on proximity to cursor when specifying a coordinate location (always uses "Normal" pick mode, even if "Query" or "Front" is selected as the pick mode)
Snap to Screen	snap to the nearest screen location when specifying a coordinate location
Snap to Grid	snap to the nearest grid location when specifying a coordinate location

Command	Description
Snap to Point	snap to the nearest point when specifying a coordinate location
Snap to Node	snap to the nearest node when specifying a coordinate location
Pick Normal	selects normal picking where closest entity is selected
Pick Query	selects all entities that are behind the cursor as you go through the depth of the model and places them in a list located lower right corner.
Pick Front	allows the selection of only the front most entity
Pick all Inside	controls how entities are selected with a box pick. If checked, the entity must be completely inside the box. If unchecked, only a part of the entity must be inside the box.
Select	This menu of options allows you to control which coordinates will be selected when you use the graphics cursor to pick a location. This is the same as capability provided with the *Cursor Position* dialog box.
Rotate View...	accesses the View Center commands (see Section 6.2.1.4, "View, Rotate, Rotate About View Center", Section 6.2.1.5, "View, Rotate, Rotate About Rotation Center...", Section 6.2.1.6, "View, Rotate, Rotate Around View Axes", Section 6.2.1.7, "View, Rotate, Rotate Around Model Axes", Section 6.2.1.8, "View, Rotate, Rotate Around Coordinate System...", Section 6.2.1.9, "View, Rotate, Rotate Around Vector...", Section 6.2.1.10, "View, Rotate, Roll-Thru...", Section 6.2.1.11, "View, Rotate, Advanced Rotate...", Section 6.2.1.12, "View, Rotate, Single Axis Rotation" in the *FEMAP Commands* manual
Equations...	calls the *Equation Editor*. This is only available when you are working in a dialog box, and in an edit or drop-down list control.
Visibility...	calls the *View, Visibility* command. This one interface allows you to control the visibility of entity types and entity labels, groups, layers, loads and constraints, and elements based on element type, element shape, or associated to materials or properties.
Post Data...	allows you to choose the output set and vectors which are used for post-processing. This is the same as the *Deformed and Contour Data* button which is available from the *View Select* command. It is not available when no output exists or when you are already in another command.

These commands are most useful in two circumstances. The first circumstance is to modify the *Snap To* setting when coordinate input is required. If a node or point exists at the appropriate coordinate location, you can change to *Snap to Node* or *Snap to Point*, select the node or point, and FEMAP will automatically use the position value as the input coordinates. *Smart Snap* may also be useful, as it will automatically select the location of a node, a point, the midpoint of a curve, or the center of a circular arc.

You could actually use the right mouse button to access the *Workplane* command, and then use the right mouse button to change the *Snap To* setting when defining the coordinates of the plane. You could even use three different methods to define the three different coordinate locations.

The other major advantage to the right mouse button is that it enables you to quickly access commands for viewing your model that are several menu commands deep. For instance, the *Visibility* command allows you to change from viewing the entire model to just viewing a group or multiple groups. If you are continuously changing the groups to view, this could become tedious to use the command from the main menu or the *View* toolbar. Instead, you can simply press *Visibility* and change to the *Group* tab. Another shortcut is to use *Post Data* (*Deformed and Contoured Data* under *View Select*) to access the *Select PostProcessing Data* dialog box.

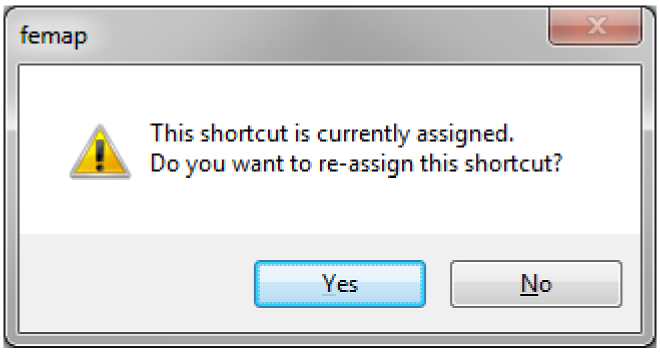
4.2.4 Shortcut Keys

FEMAP has both certain keys defined as commands for quick implementation as well as providing you the capability to define your own shortcut keys. Commands which can be accessed through standard shortcut keys have the shortcut key listed next to their name. Some of the most commonly used shortcut keys include *F5* for *View Select*, *F6* for *View Options*, and *Ctrl+Q* for *View, Visibility*. These shortcut keys enable you to access these commands without going through the menu substructure.

In addition to the standard shortcut keys, FEMAP also allows you to define letter keys in FEMAP as FEMAP commands. You can also assign currently unused function keys and keyboard combinations (i.e., CTRL, SHIFT, ALT + letter or function keys) as FEMAP commands as well. You can therefore quickly customize FEMAP to use letter and function keystrokes, as well as keyboard combinations, to represent your most often used FEMAP commands

To set up your own shortcut keys, click the “customize” triangle on any toolbar and choose the *Customize...* command or use the *Tools, Toolbars, Customize...* menu. In both cases, the *Customize...* command is at the bottom of the menu. Once in *Customize* dialog box, choose the *Keyboard* tab.

To define a shortcut key, first choose the *Category* from the drop down list, then highlight the command from the *Commands* list. After the command is highlighted, click in the “Press new shortcut key:” field and press a key or keyboard combination. Once you have chosen the correct key or keyboard combination, click the *Assign* button.



If the key or keyboard combination has already been defined, FEMAP will let you know and bring up a dialog box stating “This shortcut is currently assigned. Do you want to re-assign this shortcut?” By clicking the *Yes* button, the key or keyboard combination will be added to the “Key assignments:” list and REMOVED from the command that was previously using that shortcut key or keyboard combination. Clicking the *No* button allows you to select an unused shortcut key or keyboard combination and leaves all other shortcut keys unchanged.

Shortcut keys can be manually removed by highlighting a key or keyboard combination from the “Key Assignments:” list and then clicking the *Remove* button. The *Reset All* button will return all shortcut keys to their default commands.

Defining shortcut keys for your most used commands, you can save time moving through the FEMAP menu structure. Shortcut keys are only available from the FEMAP menu level. If you are already in another command or dialog box, pressing these keys will not have the desired effect. In most cases, it will simply result in typing the letter that you pressed. See Section 4.2.2.2, "Customizing toolbars" for some more information on creating shortcut keys.

A few of the more useful but less obvious shortcut keys are listed below. These keys work within a text or drop down list box in a FEMAP dialog box or list boxes in FEMAP. They do not apply to other Windows applications except for those noted as Windows commands. For a complete list of shortcut keys, see Section A, "Using the Keyboard".

Key(s)	Function
Ctrl+A	Measure an angle.
Ctrl+C	Copy (Windows command)
Ctrl+D	Measure a distance.
Ctrl+E	Display FEMAP Equation Editor for interactive definition of variables and equations.
Ctrl+F	List functions.
Ctrl+G	Snap cursor selections to snap grid.
Ctrl+L	Display a list of the existing entities of the desired type.
Ctrl+M	Measure a curve to attain distance
Ctrl+N	Snap cursor selections to nearest node.
Ctrl+P	Snap cursor selections to nearest point.
Ctrl+S	Snap cursor selections to screen (snap off).
Ctrl+T	Redefine snap grid.
Ctrl+V	Paste (Windows command)
Ctrl+W	Redefine workplane.
Ctrl+X	Cut (Windows command)
Ctrl+Z	Use standard coordinate selection dialog box to define location.

One of the most commonly used options is to use these keys to perform measurements when you want to input coordinates. Since these keys are available when you are in another dialog box, you can perform the measurement and obtain the result as the input to the dialog box value. There is no need to perform your measurements, write the information down, and then go into the command to define the position.

4.2.5 Status Bar

The Status bar is contained at the bottom of the FEMAP main window. By default, the left side of the Status bar keeps a running tally of the number of nodes and elements in your model. This will be overwritten by a command description if menu Help is active, but it will return when you are not accessing or pointing at a command.

In addition to the menu Help and node and element counts that appear on the left side of the bar, the right side provides one button access to:

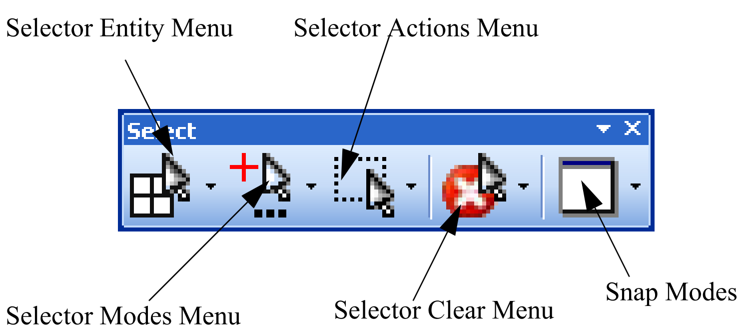
- current property
- current load set
- current constraint set
- current group (also used to choose *Show Full Model*, *Show Active* group, or *Show Multiple* groups)
- current output set

The current property, load set, constraint set, group, and output set can be changed, or a new one created, by left-clicking the mouse. Left-clicking will bring up a drop-down menu that will contain a list of the current entities or sets in the model that you can choose to activate, or you can choose *Create/Manage...* to access the *"Set Type" Manager* dialog box for the four set based items, or a simple selection box to change the current property.

The *Group* option has the added feature to toggle displaying between the full model, the active group, or multiple groups. Thus, not only can you use this feature to rapidly switch between groups when only viewing the Active group, you can also toggle between displaying the active group, multiple groups, or the entire model as a short cut to using the *Group* tab on the *View, Visibility* command (or right mouse button *Visibility* command).

4.2.6 The Select Toolbar

The *Select Toolbar* allows you to select entities one at a time or create a list of selected entities that will remain active until you toggle off or clear the selection list. This functionality allows you to choose entities of different types first and then perform multiple commands from the menus or the toolbars on the selected entities. To make the *Select Toolbar* visible, choose *Tools, Toolbars, Select.*



Selecting entities with the *Select Toolbar* is somewhat different than using dialog boxes, even though the two entity selection approaches share many of the same capabilities. The main difference is the *Select Toolbar* is designed to be used BEFORE any specific commands are selected. The *Select Toolbar* is also essential when using the dockable panes, especially the *Entity Editor* and *Data Table,* because it is often the most efficient method to place entities into either of these panes. Again, it was designed with this functionality in mind, so take advantage of the *Select Toolbar's* capabilities.

When an entity type is active for selection, you can access a context sensitive menu by clicking the right mouse button in the graphics window. Each context sensitive menu contains a set of frequently used commands for the selected entity type. These context sensitive menus can be used to help you model more efficiently.

Finally, the *Select Toolbar* has a mode known as *Show Tooltips* which allows you to query the entities in your model in a dynamic manner by simply turning the option on and highlighting entities for selection. When activated, a Tooltip note will pop up with useful information about the entity which is currently highlighted. This option can be toggled on and off and is very helpful in making sure you have the right element selected for selection or probing specific nodes and elements during post-processing.

For more information, see Section 7.3.1.6, "Tools, Toolbars, Select" and Section 5.11.3.1, "Show Tooltips"of the *FEMAP User Guide*.

4.2.7 Context Sensitive Menus

There are many Context Sensitive menus in FEMAP. A Context Sensitive menu appears when the right mouse button is clicked:

- inside the *Messages* window
- when a row or column header is highlighted in the *Data Table*, *Data Surface Editor*, or *Connection Editor*
- when an entity is highlighted in the *Model Info* tree
- while a particular entity type is active in the *Select Toolbar*
- anywhere in the *Toolbar Docking Area* not currently occupied by a Toolbar (bring up *Tools, Toolbars,...* menu)

The Context Sensitive menu in the *Messages* window contains general commands to help you use the dockable panes. Also, when a row is highlighted in the *Data Table*, a menu will give you the ability to show, filter, and delete rows from the table.

When an entity in the *Model Info* tree is selected, you can right mouse click on the selected entity and a Context Sensitive menu will appear for that particular entity type. These Context Sensitive menus provide a quicker path to many frequently used commands for the specific entity type.

While a certain entity type is active in the *Select Toolbar*, only that entity type will be available for picking in the graphics window. Since FEMAP is only highlighting one specific entity type at a time, there are context sensitive menus for each entity type. These menus can be accessed by highlighting an entity and then clicking the right mouse button. These Context Sensitive menus contain frequently used commands for each entity type.

Finally, you have the ability to quickly turn toolbars on and off one at a time by right clicking anywhere in the *Toolbar Docking Area* (above, below, to the left, or to the right of the graphics windows and Dockable Panes) not being occupied currently by a toolbar. The *Tools, Toolbars, ...* menu will come up and all the currently visible toolbars will be designated with a check mark.

For more information and lists of the Context Sensitive menus, see Section 7.2.12, "Tools, Other Windows, Messages", Section 7.2.9, "Tools, Data Table", Section 7.2.7, "Tools, Connection Editor", Section 7.2.1, "Tools, Model Info", Section 7.2.11, "Tools, Programming, Program File", Section 7.2.10, "Tools, Programming, API Programming", Section 7.3.1.6, "Tools, Toolbars, Select", and Section 7.4, "Other FEMAP Tools".

4.2.8 Dockable Pane Icons

Many of the dockable panes contain icons to perform specific commands used by the particular panes. These commands are often not available through the menu structure, and only deal with what is found in the particular dockable pane.

For more information about the dockable pane icons, see Section 7.2.1, "Tools, Model Info", Section 7.2.2, "Tools, Meshing Toolbox", Section 7.2.3, "Tools, PostProcessing Toolbox", Section 7.2.4, "Tools, Charting", Section 7.2.5, "Tools, Entity Editor", Section 7.2.6, "Tools, Data Surface Editor", Section 7.2.7, "Tools, Connection Editor", Section 7.2.8, "Tools, Entity Info", Section 7.2.9, "Tools, Data Table", Section 7.2.10, "Tools, Programming, API Programming", and Section 7.2.11, "Tools, Programming, Program File".

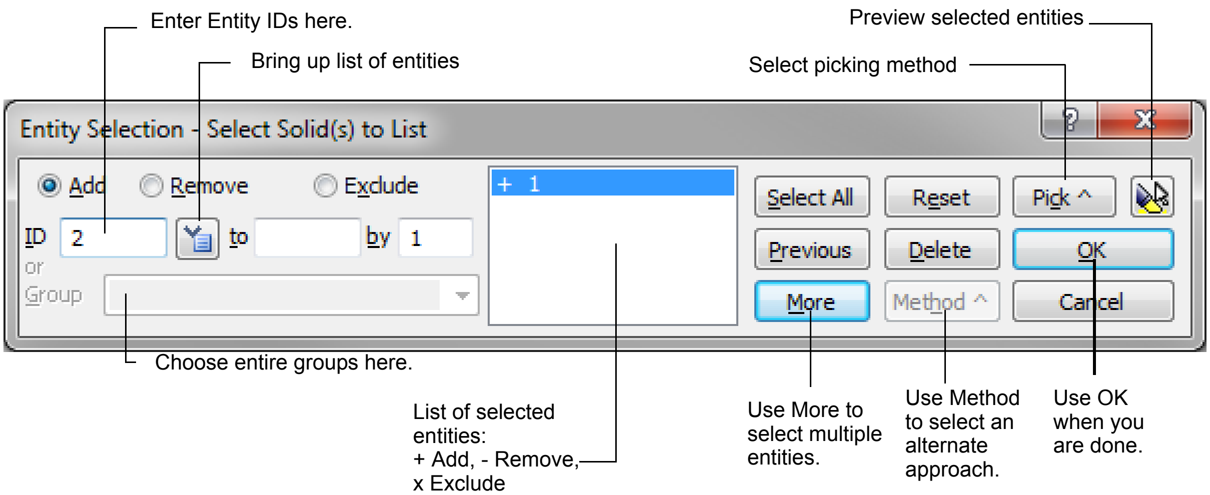
4.3 FEMAP Dialog Boxes

Many FEMAP commands require additional input to control its actions. Some commands require you to select geometry or other entities. Others require you to specify coordinates or choose from a list of available options. In all of these cases, one or more dialog boxes is displayed to request and accept that input.

4.3.1 Entity Selection

In FEMAP, just as in other Windows applications, when a dialog box is displayed you are presented with one or more options, or requests for text/numeric input. To select an option, or specify some input, you first move to that field/control using either the keyboard (*TAB*, *Direction* or *Alt*+underlined_letter keys) or more directly by selecting it with the cursor. If the current control is a button or list box, you can select an option using either the keyboard or the cursor. If the current control is a text box or drop-down list box many applications give you no choice but to go back to your keyboard and type your input. With FEMAP, you can still do that, but in most cases (IDs and coordinates) you can also enter the text/numeric input graphically using the cursor.

FEMAP has several standard dialog boxes, but the most common box is the *Entity Selection* dialog box shown below. Many FEMAP commands require you to select one or more entities which will be used for the command. For all commands which support the selection of multiple entities, FEMAP uses a common entity selection dialog box. With this dialog box you can select any combination, and any number of entities from your model. As entities are selected or removed, they appear in the list of selected entities which is located near the center of the dialog box



Since this box appears whenever you need to select entities for the command you have chosen, it is very important that you become familiar with this dialog box. You will see some form of it over and over again. A brief explanation of each feature is provided below.

Choosing a Picking Method

Details regarding the entity selection box are provided below, however the most important things to remember are:

- how to orient the model
- picking methods
- using alternative methods to speed picking

Orienting the Model

When the *Entity Selection* dialog box is open, you may need to rotate, pan, or zoom the model to get a better view while you are picking entities. You can use the middle mouse button (or wheel) to dynamically view the model. You can:

- rotate by pressing the middle mouse button
- pan by pressing the middle mouse button and holding the *Ctrl* key
- zoom by pressing the middle mouse button and holding the *Shift* key or spinning the wheel of a Wheel mouse

If you have a two-button mouse, you can use the *Dyn Rotate* icon on the *View Toolbar* instead.

Picking Methods

The *Pick* button allows you to access many different types of picking including *Normal, Query, Front, Box, Circle, Polygon, Freehand*, *Coordinate*, *Around Point*, *Around Vector*, and *Around Plane* picking. There are several modes for picking *Combined Curves* and *Boundary Surfaces* which act like “filters” to determine what can be selected when *Combined Curves* and/or *Boundary Surfaces* exist in your model. There are also two commands on this menu, *Add Connected Fillets* and *Add Tangent Surfaces,* which first require you to choose an entity, then add more entities of that kind based on the specified criteria.

By default, the entity selection box allows you to select entities in the graphics windows one by one. To select all entities inside a box, select *Box* (or alternatively hold down the *Shift* key, and then press and hold down the left mouse button). You can now drag the cursor on screen to select all entities within a rectangle. Alternatively, using *Circle* (or the *Ctrl* key instead of the *Shift* key) will circle pick.

The *Query* pick allows you to bring up a list of entities that have a similar XY screen location thus allowing you to better understand what is being selected. *Query* can be accessed temporarily by holding down the *Alt* key while clicking. The *Front* mode selects only the entities that are in the front of the model.

The *Polygon* and *Freehand* options are just what they suggest. The polygon option allows you to pick points for a polygon pick, while freehand allows you to literally draw on the screen.

The *Coordinate* option allows you to select entities using a combination of X,Y, and/or Z values referencing a selected coordinate system along with various limiting criteria (Above or Below a single value; Between or Outside two values; or At Location, within a specified Tolerance).

The *Around Point* and *Around Vector* options allow you to select entities using each entity's position in 3-D space in relation to a specified "definition entity" (Specified Point in 3-D space or Specified Vector) along with various limiting criteria (Farther Than or Closer Than a single value; Between or Outside two values; or At Location, within a specified Tolerance).

The *Around Plane* option allows you to select entities using each entity's position in 3-D space in relation to a Specified Plane along with various limiting criteria (Positive Side or Negative Side of Plane with offset value; Between or Outside two offset values; or At Location, within a specified Tolerance).

The *Model Data Value* option allows you to select entities in the model which all use a specific material/property value or have values which fall within a range of values for a particular material/property entry (i.e., Plane Element Thickness, Young's Modulus, BEAM End A Area etc.).

The *Color* option allows you to select a color from the *Color Palette*, then adds all entities of the current type which are also that color to the selection list.

Add Connected Fillets is a "one time" command which will add all of the "connected fillets" to any number of selected surfaces representing fillets in your model. In a similar manner, *Add Tangent Surfaces* will add all surfaces tangent to any number of surfaces already selected to the selection list. Finally, *Add Connected Elements* will add all elements connected by at least one node to any of the elements already selected, regardless of element type.

Using Alternative Methods to Speed Picking

Individually picking each entity, or even multiple picking options are not always the most efficient method to select FEMAP entities. Familiarize yourself with the various methods that are available via the *Methods* button in the entity selection dialog box. For example, when selecting elements, there are methods that make it very easy to select; all elements referencing a node, all element referencing a certain material, all elements of a certain type, all elements on a surface, etc.

Entity Selection Options

Add, Remove, Exclude:

These options control whether the next entity will be added to, or subtracted from, the list of selected entities. The default is always to add entities to the list. The *Add* and *Remove* options are order dependent. If you remove an entity, and then later add it again, the entity will be included in the list since the add occurred last. The *Exclude* option is the same as *Remove*, except that it is not order dependent. If you exclude an entity, and then later add it, it will not be included in the list no matter how many times you attempt to add it. Any of the options can be chosen any number of times, even for the same entity. For example, you can add the same entity 10 times if you want, although it will be treated just as if you had added the entity one time.

When entities are shown in the selection list, the first character indicates whether that entry adds, removes or excludes the entity. All added entities will be preceded by a +. "Remove" selections are indicated by a - and "Exclude" selections by an x.

ID, to, by:

These three text boxes are the primary input controls. In many cases, you will simply want to select a single entity. In this case, just enter the entity's ID into the ID text box. If you want to select a range of entities, enter the minimum (into ID) and maximum ID (into To), and the increment (most often 1).

Select From List Button :

For entities which can have "Titles" (includes Solids, Coordinate Systems, Materials, Properties, Layups, Load Sets, Constraint Sets, Connection Properties, Regions, Connectors, Functions, Analysis Sets, Groups, Views, Aero Panel/Bodies, Aero Properties, Aero Splines, Aero Control Surfaces, Freebody entities, and Output Sets) the "Select from List" button can be used to choose "Titled" entities from a "Multi-select" dialog box. Any number of existing entities can be selected from the list by checking them individually or highlighting the titles and using the *Selected On* icon button to check multiple items. Additional icon buttons exist to perform *All On*, *All Off*, and *Selected Off* operations. The number of items in the list can be reduced using a "matching text" filter by entering text in the field above the list, then clicking the *Filter* icon button to show only items containing the entered text. Simply click the *Clear Filter* icon button to restore the full list.

Note: This command differs slightly from clicking Ctrl+L in a dialog box field to bring up a list of "Titled" entities, as only one entity at a time can be selected using that method.

Group:

If you have defined one or more groups in your model (using the *Group* menu) you can use them to quickly identify the list of entities to be selected. Use the drop-down list to view all of the available groups. If you choose *More*, all entities from the group will be loaded into the list of selected entities.

Type (Type and Shape methods only):

The ID fields will be replaced by a drop-down list of available Types (Coordinate System, Element, Material, and Property Types) or Shapes (Element Topologies).

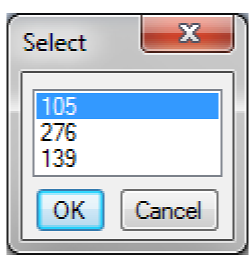
Pick:

The *Pick* button provides access to various methods of graphical selection. A menu will appear with various options.

The **Normal** option simply allows you to select one entity at a time from the screen. The other options provide for multiple entity selection and are explained more fully below. It is important to note that the *Box* and *Circle* picking options can be accessed in Normal mode by holding down the *Shift* and *Ctrl* keys, respectively, clicking and holding the left mouse button, and dragging the cursor across the graphics screen.

Query

This option selects all entities that are behind the cursor as you go through the depth of the model and places the IDs in a list located in the lower right corner of the screen by default. When an entity ID is selected in the list, the associated entity will be highlighted in the Graphics window allowing you to distinguish between coincident or nearly coincident entities.



You can scroll the list in three ways, using the up down arrow keys, the roller on the mouse, or clicking the right mouse button to move to the next ID in the list. When the entity you wish to select is highlighted you can select the left mouse button or press *OK* in the *Query* list box.

Note: You can either turn on the *Query* mode by selecting it from the *Pick* menu, or you can use it for a single pick by simply holding down the *Alt* key while clicking. When you release the *Alt* key, the picking mode will return to its previous state (either *Normal* or *Front*).

Front

This option also uses the depth of the model, but instead of bringing up a list like *Query*, it only allows you to select the entity that is "closest" to you. Once an entity is chosen, the one behind it becomes available to pick and so on.

Box Picking

If you select this option, simply click on the left mouse button at one end of the box, drag the cursor to the other end of the box, and release the mouse button. This will select all entities inside the box. FEMAP provides a graphical preview of the box when you drag the cursor. If you do not want to select any entities in the box, press *Cancel*.

Circle Picking

This option works much like box picking except the original click of the left mouse button defines the center of the circle (instead of the corner of the box), and the location at which you release the button is a point on the circle.

Polygon Picking

This option is an extension of the box picking, except instead of holding the left mouse button down, you click on specific locations. FEMAP will create a polygon from click to click for the entity selection. You can press *Done* after your last location, or double click the last point, or close the polygon by repicking the first location (the dotted line changes to solid when you are over the first location).

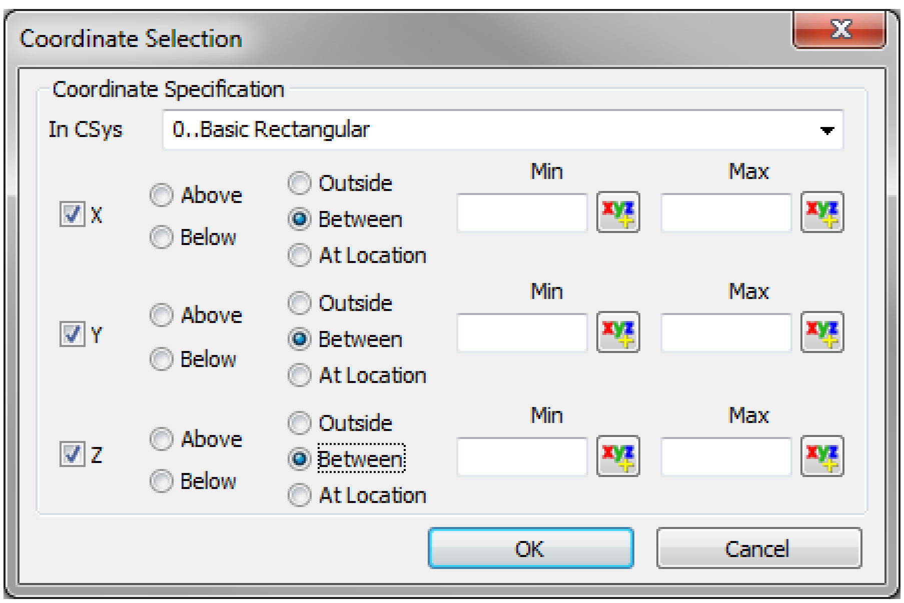
Freehand Picking

This option provides the most flexible input. Simply drag the cursor by holding the left mouse button down. When you have completed the area you want to select, simply release the button. FEMAP will then automatically select those entities in your freehand sketch.

Note: The picking method always returns to *Normal* after you have performed a picking procedure. If you need to create another polygon or freehand sketch for picking, simply select this option again under *Pick*.

Coordinate Picking

The *Coordinate* option allows you to select entities using a combination of X,Y, and/or Z values referencing a selected coordinate system along with various limiting criteria (Above or Below a single value; Between or Outside two values; or At Location, within a specified Tolerance).



You can choose any coordinate system in your model and then select X, Y, and/or Z and a "limiting criteria" for each coordinate. You can click the "Graphical Pick" Icon button next to any active field and this allows you to get a value for that field by graphically picking in the model.

When using the *At Location* criteria, a "Tolerance" is used and can be manually entered. By default, this value is set to the "Merge Tolerance" of your model and "expands" the selection area +/- that value (See Section 7.4.1, "Tools, Parameters..." for how "Merge Tolerance" can be defined). You can also enter a larger value to "expand" the selection area further in both directions.

Any value entered in a field as selection criteria WILL be included in the selection.

For example, say you want to list all nodes with an X coordinate "above" a value of "1.0" in the Global Rectangular Coordinate System in your model. In order to do this, check the box next to *X* to make it "active" (make sure the

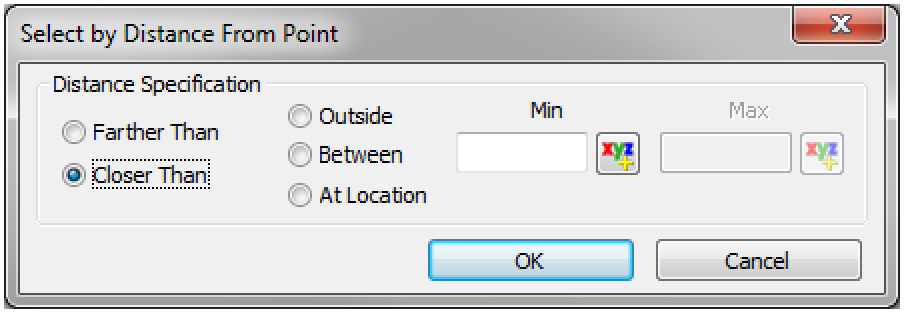
Y and *Z* boxes are "unchecked"), choose the *Above* criteria, then enter a value of "1.0" into the *Max* field. When you click OK, ALL nodes with an X value of "1.0" AND Above will be selected.

If you do not want the nodes at "1.0" to be included in the selection, you would want to enter a slightly higher value (i.e.,"1.000001") OR use an "mathematical operator" to slightly increase the value (i.e.,"1.0+1E-8").

Around Point

The *Around Point* option allows you to select entities using each entity's position in 3-D space in relation to a "Specified Point" along with various limiting criteria (Farther Than or Closer Than a single value; Between or Outside two values; or At Location, within a specified Tolerance). Essentially, a "sphere" will be created around the "Specified Point" and selection will be based on the defined limiting criteria.

FEMAP will first prompt you for a point using the standard *Locate* dialog box and any "coordinate definition method" can be used. Once the "Point" has been specified, the *Select by Distance From Point* dialog box will appear.



You can click the "Graphical Pick" Icon button next to any active field and this allows you to get a value for that field by graphically picking in the model.

When using the *At Location* criteria, a "Tolerance" is used and can be manually entered. By default, this value is set to the "Merge Tolerance" of your model and "expands" the selection area +/- that value (See Section 7.4.1, "Tools, Parameters..." for how "Merge Tolerance" can be defined).

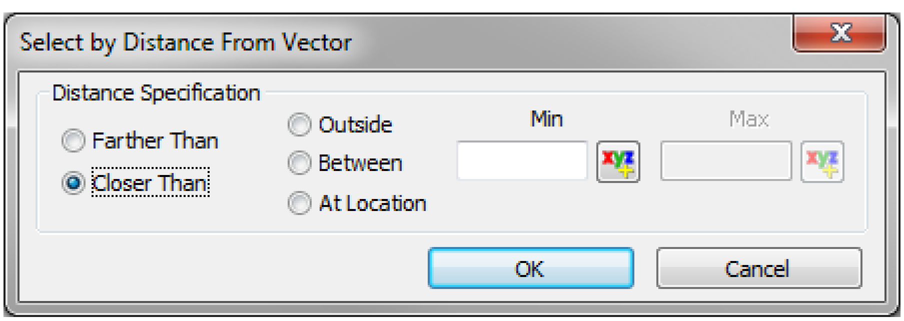
Any value entered in a field as selection criteria WILL be included in the selection.

For example, say you want to list all nodes closer than "1.0 unit" away from a specified point in space. In order to do this, choose the *Closer Than* criteria, then enter a value of "1.0" into the *Min* field. When you click OK, ALL nodes within a "1.0 unit" sphere AND any nodes exactly "1.0 unit" in any direction will be selected.

If you do not want the nodes at "1.0 unit" to be included in the selection, you would want to enter a slightly lower value (i.e.,".9999999") OR use an "mathematical operator" to slightly increase the value (i.e.,"1.0-1E-8").

Around Vector

The *Around Vector* option allows you to select entities using each entity's position in 3-D space in relation to a "Specified Vector" along with various limiting criteria (Farther Than or Closer Than a single value; Between or Outside two values; or At Location, within a specified Tolerance). Essentially, a "cylinder" will be created around the "Specified Vector" and selection will be based on the defined limiting criteria.



FEMAP will first prompt you for a vector using the standard *Vector Locate* dialog box and any "vector definition method" can be used. Once the "Vector" has been specified, the *Select by Distance From Vector* dialog box will appear.

You can click the “Graphical Pick” Icon button next to any active field and this allows you to get a value for that field by graphically picking in the model.

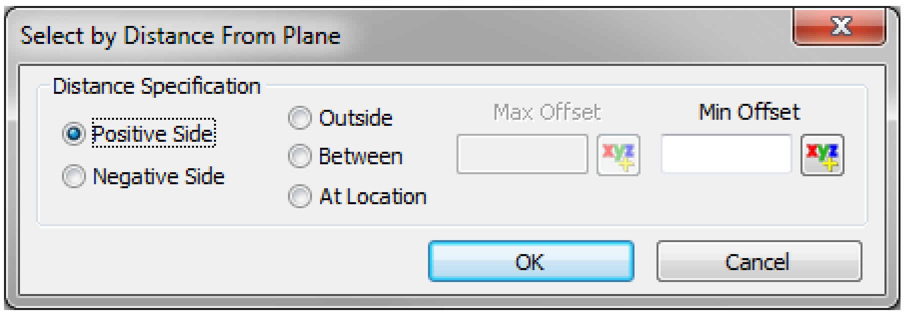
When using the *At Location* criteria, a “Tolerance” is used and can be manually entered. By default, this value is set to the “Merge Tolerance” of your model and “expands” the selection area +/- that value (See Section 7.4.1, "Tools, Parameters..." for how “Merge Tolerance” can be defined).

Any value entered in a field as selection criteria WILL be included in the selection.

For example, you want to list all nodes farther than “1.0 unit” away from a specified vector. To do this, choose the *Farther Than* criteria, then enter a value of “1.0” into the *Max* field. When you click OK, ALL nodes outside a “1.0 unit” cylinder AND any nodes exactly “1.0 unit” away from the vector in the radial direction will be selected. If you do not want the nodes at “1.0 unit” to be included in the selection, you would want to enter a slightly higher value (i.e.,“1.000001”) OR use an “mathematical operator” to slightly increase the value (i.e.,“1.0+1E-8”).

Around Plane

The *Around Plane* option allows you to select entities using each entity’s position in 3-D space in relation to a “Specified Plane” along with various limiting criteria (Positive Side or Negative Side of Plane with offset value; Between or Outside two offset values; or At Location, within a specified Tolerance).



FEMAP will first prompt you for a plane using the standard *Plane Locate* dialog box and any “plane definition method” can be used. Once the “Plane” has been specified, the *Select by Distance From Plane* dialog box will appear.

The “Positive Side” is the side of the “Specified Plane” with the “positive normal direction” (based on the right hand rule) and the other side is the “Negative Side”. You can enter an Offset Distance from the plane in either the Positive or negative direction.

You can click the “Graphical Pick” Icon button next to any active field and this allows you to get a value for that field by graphically picking in the model.

When using the *At Location* criteria, a “Tolerance” is used and can be manually entered. By default, this value is set to the “Merge Tolerance” of your model and “expands” the selection area +/- that value (See Section 7.4.1, "Tools, Parameters..." for how “Merge Tolerance” can be defined).

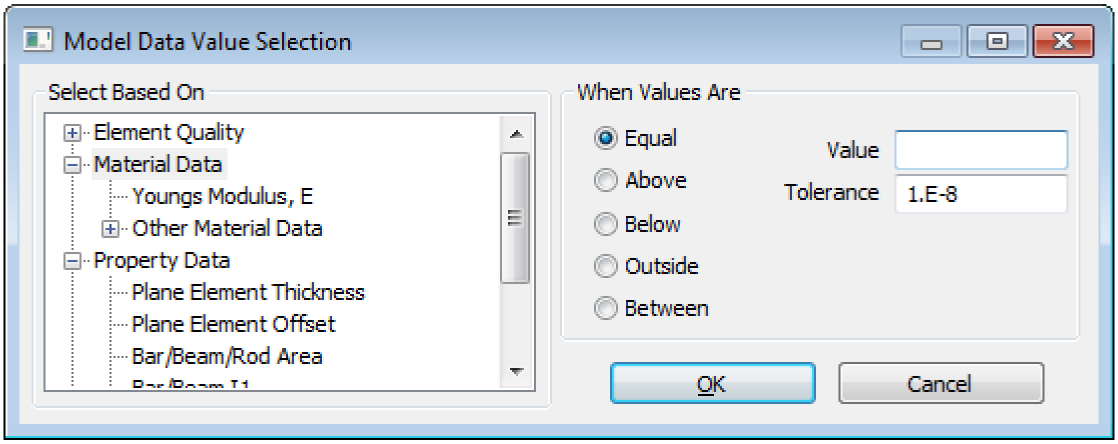
Any value entered in a field as selection criteria WILL be included in the selection.

For example, say you want to list all nodes between “-1.0 Unit” and “+1.0 Unit” offset from the specified plane. In order to do this, choose the *Between* criteria, then enter a value of “-1.0” into the *Min* field and “1.0” into the *Max* field. When you click OK, ALL nodes between “-1.0 unit” and “+1.0 Unit” from the plane AND any nodes exactly “+/-1.0 unit” away from the plane will be selected.

If you do not want the nodes at “1.0 unit” +/- the plane to be included in the selection, you would want to enter a slightly lower value (i.e.,“+/- 0.999999” in the appropriate fields) OR use an “mathematical operator” to slightly increase the value (i.e.,“1.0-1E-8”).

Model Data Value

The *Model Data Value* option allows you to choose entities in the model with values *Equal* to a specific element quality value or material/property value (i.e., Plane Element Thickness, Young’s Modulus, BEAM End A Area etc.) or have values within a range (*Above* or *Below* a single value; *Between* or *Outside* two values) for a particular material/property entry.



When using the *Equal* criteria, a "Tolerance" is used and can be manually entered. By default, this value is set to the 1.0E-8 and "expands" the selection area +/- that value

Any value entered in a field as selection criteria WILL be included in the selection.

Color

The *Color* option allows you to select a color from the *Color Palette*, then adds all entities of the current type which are also the selected color to the selection list. Options also exist to *Match Color*, *Match Pattern/Transparency*, and *Match Line Style* options which may turned on/off to either broaden or narrow the selection criteria. By default, all *Options* are on.

For Example, if you wanted all elements in the model which are Red (specifically Color 4), regardless of the selected *Pattern/Transparency* or *Line Style*, you would probably want to "uncheck" the *Match Pattern/Transparency*, and *Match Line Style* options if the colors in the model use any of these options.

Combined Curves pick mode

There are four modes when selecting *Combined Curves* in FEMAP, *Default*, *All Points/Curves*, *Points/Curves Eliminated by Combined Curves*, and *Combined Curves Only*. Once *Entity Selection* dialog has been closed by *OK* or *Cancel*, this pick mode always returns to *Default*. Here is a brief description of each:

- *Default* - In this mode, all "individual curves" used to create *Combined Curves* can no longer be selected. The *Combined Curves* are now available for selection along with any "individual curve" currently not being used by any *Combined Curves*. Also, the end points of any "internal" curves of a *Combined Curve* can no longer be selected.
- *All Points/Curves* - All "underlying" points and curves used by *Combined Curves* are available for selection, as well as the *Combined Curves* themselves and any "individual curves" in the model.
- *Points/Curves Eliminated by Combined Curves* - Only "underlying" points and curves used by *Combined Curves* are available for selection.
- *Combined Curves Only* - Only *Combined Curves* can be selected.

Boundary Surfaces pick mode

There are also four modes when selecting *Boundary Surfaces* in FEMAP, *Default*, *All Curves/Surfaces*, *Curves/Surfaces Eliminated by Boundary*, and *Boundary Surfaces Only*. Once *Entity Selection* dialog has been closed by *OK* or *Cancel*, this pick mode always returns to *Default*. Here is a brief description of each:

- *Default* - In this mode, all "individual surfaces" used to create *Boundary Surfaces* can no longer be selected. The *Boundary Surfaces* themselves are now available for selection along with any "individual surface" currently not being used by any *Boundary Surfaces*. Also, the "internal" curves of *Boundary Surfaces* can no longer be selected.
- *All Curves/Surfaces* - All "underlying" surfaces and curves used by *Boundary Surfaces* are available for selection, as well as the *Boundary Surfaces* themselves and any "individual surfaces" in the model.
- *Curves/Surfaces Eliminated by Combined Curves* - Only "underlying" surfaces and curves used by *Boundary Surfaces* are available for selection.

- *Boundary Surfaces Only* - Only *Boundary Surfaces* can be selected.

Add Connected Fillets

Using the *Add Connected Fillets* command allows you to quickly add “connected fillets” to the selection list by first choosing any number of surfaces which represent fillets in your geometry. This is a helpful picking tool when using the *Geometry, Solid, Remove Face* (Section 3.4.2.16, "Geometry, Solid, Remove Face...") to try and remove fillets from geometry. Only visible when selecting surfaces.

Add Tangent Surfaces

Like *Add Connected Fillets*, the *Add Tangent Surfaces* command adds surfaces based on their relationship to surfaces which have already selected. In this case, surfaces “tangent” to any number of surfaces already in your selection list will be added to the list. This is a helpful command when you would like to pick all of the surfaces on “one side” of a part. Only visible when selecting surfaces.

Add Connected Elements

Using the *Add Connected Elements* command allows you to quickly add “connected elements” to the selection list by first choosing any number of elements. Any element which is connected by at least one node to the already selected elements will be added. Only visible when selecting elements.

Add All Connected Elements

Using the *Add Connected Elements* command allows you to quickly add all of the “connected elements” to the selection list by first choosing any number of elements. Any element which is connected by at least one node to the already selected elements will be added, then any element connected by at least one node to those elements will also be added, until there are no more connected elements to add. Only visible when selecting elements.

By Faces

Picking *By Faces* is only available when you are selecting Nodes or Elements. It provides access to the face selection methods described in the *Model, Load, Elemental* command, including selection of Adjacent Faces. When you use this method of selection during Node selection, all nodes on the element faces that you pick are selected. During element selection, the elements containing the faces are selected.

By Output

Like *By Faces*, this method is only available when picking nodes or elements, and only if you have loaded analysis results into your model. Usage of this selection method is just like the *Group, Operations, Generate With Output* command, where you can find more information. In this case however, the selection is not used to create a group, rather it is directly added to the selection dialog box.

Copy, Copy as List, and Paste

In some cases, you may want to transfer selection lists (the IDs being selected) to other programs. By choosing *Copy*, the IDs that have been selected (already in the list) will be copied to the Clipboard, and will be available to be pasted as text into another application. If there are any IDs which have been removed or excluded from the selection list (designated with a “-” or a “x” before the ID or an ID range), they will NOT be copied to the clipboard. The format on the Clipboard is a simple, *startID, stopID, increment* - just as it is shown in the list.

Copy as List uses the same methodology as *Copy* for choosing which IDs will be copied to the clipboard, but will send only the IDs (no range notation) to the clipboard. For example, you select a range of elements which appear in the dialog box like this: “+1,5,1” (using the *startID, stopID, increment* format). By choosing *Copy as List*, FEMAP will copy this range to the clipboard as individual entities (1, 2, 3, 4, 5) instead of using the range format.

Pressing *Paste* is the opposite of *Copy.* A list of IDs is copied from the Clipboard and then added, removed, or excluded from the selection list depending on the current mode selected in the dialog box. When the mode is set to *Add*, the entities will have a “+” in front of the IDs after being pasted into the dialog box, when set to *Remove* they will be designated with a “-”, and when set to *Exclude* with a “x”. You can use these commands together to *Copy* a selection into another application, edit the IDs, then *Paste* it back into the selection dialog.

Preview Button: 

Use this option to highlight - like the *Window, Show Entities* command - all of the entities that you have picked so far. Each of the entities that you have placed into the selection list will be highlighted on the screen. After previewing your selection, you can change your selection and preview again. The color and style of highlighting are controlled by your current settings in the *Window, Show Entities* command. If you want to change them, simply go to that command, pick a new color or new options, and they will be used for future previews.

Preview is only available for the same entity types that are available in *Window, Show Entities*. When selecting other entities, *Preview* will be disabled. If you are not using ID selection, but have switched to some other method, you will see the selection list go blank when you press *Preview.* Your entities are still selected; they have simply been converted to an ID list - just like they would be if you switched to a new method. You can continue to select using this method, but if you want to remove a selection, you must switch to *Exclude* mode.

Select All:

Choosing this button selects all entities of the desired type. The selection mode is independent of the *Add, Remove, Exclude* options. The entities are always added. You will see a single entry in the list of selected entities which looks like:

```
+ minID,maxID,1
```

where minID and maxID are the minimum and maximum entity IDs respectively. Do not worry if you have gaps in your numbering, FEMAP will only choose existing entities between (and including) minID and maxID.

Previous:

Whenever you complete a selection and press *OK*, FEMAP remembers the list of selected entities. The next time that you need to select entities of the same type, you can choose this button to reuse your previous selections. The previous IDs are placed into the selection list depending on which mode, *Add* (“+”)*, Remove* (“-”)*,* or *Exclude* (“x”)*,* is currently selected in the dialog box. A separate list is saved for each type of entity, but the appropriate list is overwritten every time the *Entity Selection* dialog box is displayed and you choose *OK*.

Reset:

If you have already made some selections, this will erase all of them and start over. The entity selection list will be blanked.

Delete:

This is a more selective version of *Reset* which allows you to edit the list of selected entities. First, select the entry in the list that you want to eliminate, either by using the *Tab* key to move to the list, then using *Up* or *Down* to make the selection, or more simply by clicking on the entry with the mouse. Then choose *Delete* - the selected entry will be removed from the list, and those entities will no longer be selected (or deselected if the entry you deleted was a remove/exclude entry).

Method:

The *Method* button will enable you to change the way entities are selected. When the dialog box first appears, you will always be selecting entities by their ID. If you press the *Method* button, you will see a popup menu that contains additional ways to select entities. For example, if you are choosing elements, you will be able to choose elements by selecting them by their *ID, Material, Property,* the *Type of Element*, or even based upon the nodes used.

You may even select one method, choose the desired entities, switch methods, and add additional entities. FEMAP will automatically choose the ID of the elements which are referenced by these other entities and place them in the selection box under the ID method. All operations such as *Add, Remove,* and *Exclude* are still applicable even when mixing the *Methods* selection. A list of the available methods for the applicable entities are provided below.

Entity	Rule / Command	What You Define	What is Selected
Point	ID	Point IDs	IDs you select.
	Color	Point ID	All points with same color as a selected point
	Layer	Point ID	All points on same layer as a selected point
	Property	Property ID	All points with selected property as a mesh attribute
	Definition CSys	CSys IDs	Any points defined relative to IDs you select.
	on Curve	Curve IDs	Any point used to define a selected curve.

Entity	Rule / Command	What You Define	What is Selected
Curve	ID	Curve IDs	IDs you select.
	Color	Curve ID	All curves with same color as a selected curve
	Layer	Curve ID	All curves on same layer as a selected curve
	in Region	Region IDs	Any curves referenced by a selected region
	Property	Property ID	All curves with selected property as a mesh attribute
	using Point	Point IDs	Any curve which references a selected point.
	on Surface	Surface IDs	Any curve used to define a selected surface.
	on Solid	Solid ID	Any curve used to define a selected solid
Surface	ID	Surface IDs	IDs you select.
	Color	Surface ID	All surfaces with same color as a selected surface.
	Layer	Surface ID	All surfaces on same layer as a selected surface
	in Region	Region IDs	Any surfaces referenced by a selected region
	Property	Property ID	All surfaces with selected property as a mesh attribute
	using Curve	Curve IDs	Any surface which references a selected curve.
	on Volume	Volume IDs	Any surface used to define a selected volume.
	on Solid	Solid ID	Any surface used to define a selected solid
Solid	ID	Solid IDs	IDs you select.
	Layer	Solid IDs	All solids on same layer as a selected solid(s)
	Property	Property IDs	All solids with selected properties as a mesh attribute
	Type	Solid Types	Any solid of a selected type.
	using Curve	Curve IDs	Any solid which references a selected curve.
	using Surface	Surface IDs	Any solid which references a selected surface.
Volume	ID	Volume IDs	IDs you select.
	Color	Volume ID	All volumes with same color as a selected volume.
	Layer	Surface ID	All volumes on same layer as a selected volume
	Property	Property ID	All volumes with selected property as a mesh attribute
	using Surface	Surface IDs	Any volume which references a selected surface.
Connection Property	ID	Connector ID	IDs you select.
	Color	Connector ID	All connection properties with same color as a selected connector(s)
	Layer	Connector ID	All connection properties on same layer as a selected connector(s)
	On Connector	Connector ID	All connection properties used by selected Connector(s)
Region	ID	Region IDs	IDs you select.
	Color	Region ID	All Regions with same color as a selected Region
	Layer	Region ID	All Regions with same layer as a selected Region
	on Connector	Connector ID	All Regions used by selected Connector(s)
	on Solid	Solid ID	All Regions which reference a Surface, Curve, Node, or Element on or associated to the selected Solid(s)
	referencing Node	Node ID	All Regions defined using selected Node(s) OR the regions which will include selected Node(s) when “expanded for export” to a solver
	referencing Element	Element IDs	All Regions defined using selected Element(s) and/or Faces of Element(s) OR the Regions which will include selected Element(s) and/or Faces of Element(s) when “expanded for export” to a solver
	using Curve	Curve ID	All Regions defined using selected Curve(s)
	using Surface	Surface ID	All Regions defined using selected Surface(s)
	using Property	Property ID	All Regions defined using selected Property(s)

Entity	Rule / Command	What You Define	What is Selected
Connector	ID	Connector ID	IDs you select.
	Color	Connector ID	All connectors with same color as a selected connector.
	Layer	Connector ID	All connectors with same color as a selected connector.
	Property	Connector ID	All connectors with same connection property as a selected connector
	using Region	Region IDs	Any connector using the selected Region(s)
Coordinate System	ID	CSys IDs	User-defined Csys IDs you select.
	Color	CSys IDs	All User-defined Csys with the same color as the selected Csys.
	Layer	Csys IDs	All User-defined Csys on the same layer as the selected Csys.
	Definition CSys	CSys IDs	Any User-defined CSys defined relative to IDs you select.
	Type	CSys Types (0,1,2)	Any User-defined Csys of selected type.
	on Point	Point IDs	All User-defined Csys located at a point
	on Node	Node IDs	All User-defined Csys located at a node
	on Element	Element IDs	All User-defined Csys used by the selected element
	on Property	Property IDs	All User-defined Csys used by the selected property
	on Csys	CSys IDs	All User-defined Csys used by the selected Csys as definition Coordinate System
Node	ID	Node IDs	IDs you select.
	ID - Free Edge	Node IDs	IDs you select but only those on free edges
	ID - Free Face	Node IDs	IDs you select but only those on free faces
	ID - Constrained	Node IDs	IDs you select but only those that are constrained
	ID - Constraint Equation	Node IDs	IDs you select but only those that are attached to constraint equations
	ID - Loaded	Node IDs	IDs you select but only those that have loads
	Color	Node IDs	All nodes with same color as a selected node
	Layer	Node IDs	All nodes with same layer as a selected node
	Definition CSys	CSys IDs	Any node defined relative to IDs you select.
	Output CSys	CSys IDs	Any node with output CSys equal to IDs you select.
	on Element	Element IDs	Any node used to define a selected element.
	Element Orientation	Element IDs	Any node used to define a selected element’s orientation (i.e., bars and beams)
	Superelement ID	Node IDs	Any node having the same Superelement ID as the selected nodes
	in Region	Region IDs	Any nodes referenced by a selected region
	on Point	Point ID	Any node which references a selected point
	on Curve	Curve ID	Any node which references a selected curve
	on Surface	Surface ID	Any node which references a selected surface
	in Solid/Volume	Solid/Volume ID	Any node which references a selected solid/volume

Entity	Rule / Command	What You Define	What is Selected
Element	ID	Element IDs	IDs you select.
	ID - Free Edge	Element IDs	IDs you select but only those with free edges
	ID - Free Face	Element IDs	IDs you select but only those with free faces
	ID - Loaded	Element IDs	IDs you select but only those that have loads
	Color	Element IDs	All elements with same color as a selected element
	Layer	Element IDs	All elements with same layer as a selected element
	Material	Material IDs	Any element which references a material (via a property) you select.
	Property	Property IDs	Any element which references a property you select.
	Layup	Layup IDs	Any element using the selected layup(s)
	Type	Element / Property Types	Any element of a selected type.
	Shape	Element Shape	Any element with same shape as the selected element.
	using Node	Node IDs	Any element which references a selected node.
	All Nodes	Node IDs	Any element for which ALL nodes used to define that element have been selected.
	in Region	Region IDs	Any elements referenced by a selected region
	on Point	Point ID	Any element which references a selected point
	on Curve	Curve ID	Any element which references a selected curve
	on Surface	Surface ID	Any element which references a selected surface
	in Solid/Volume	Solid/Volume ID	Any element which references a selected solid/volume
Material	ID	Material IDs	IDs you select.
	Color	Material IDs	All materials with same color as a selected material
	Layer	Material IDs	All materials with same layer as a selected material
	on Property	Property IDs	Any material which is referenced by a selected property.
	on Element	Element IDs	Any material which is referenced (via a property) by a selected element.
	Type	Material Types	Any material of a selected type.
Property	ID	Property IDs	IDs you select.
	Color	Property IDs	All properties with same color as a selected property
	Layer	Property IDs	All properties with same layer as a selected property
	on Element	Element IDs	Any property which is referenced by a selected element.
	Material	Material IDs	Any property which references a selected material.
	Layup	Layup IDs	Any property which references a selected layup
	Type	Element / Property Types	Any property of a selected type.
	in Region	Region IDs	Any property referenced by a selected region
	on Point	Point ID	Any property used as a meshing attribute on a selected point
	on Curve	Curve ID	Any property used as a meshing attribute on a selected curve
	on Surface	Surface ID	Any property used as a meshing attribute on a selected surface
	in Solid/Volume	Solid/Volume ID	Any property used as a meshing attribute on a selected solid

The *Method* button appears not only in the *Entity Selection* dialog box, but also in most standard dialog boxes, such as defining a coordinate location, a vector, or a plane. There are currently 18 methods available to define a coordinate location, 13 methods to define a vector, and 11 methods to define a plane. If you need to define a point, vector, or plane, and you think there is an easier method than simply inputting the coordinates of the locations, there probably is. Check the *Methods* button to see what options are available to you. It can save you tremendous amounts of effort by using different methods.

Hint: The method's ID-constrained and ID-loaded dialog boxes set up rules that allow you to only select entities related to a specific load or constraint. First select the filter you wish to use, then use the graphical selection methods such as box pick to select a large area of the model. FEMAP will then apply the filter to all of the entities in the box and only select those entities that pass the filter.

More, OK:

These options select the entities specified by ID, to, by or the entities in the selected group. The entities will be included in the selection list based on the setting of *Add, Remove, Exclude*. The only difference between *More* and *OK* is that *OK* finishes your selection while *More* lets you select additional entities or simply review the list.

4.3.1.1 Graphical Selection

One of the most powerful features of the entity selection dialog box is its ability to select entities graphically. Before you can select entities graphically, **you must make sure that the keyboard focus is set to the ID field, just as if you were going to type an ID**. This is always the case when the dialog box is first displayed. You can check however by looking for the blinking vertical bar cursor. If it is in the ID field you are ready to go, otherwise click with your mouse in the ID field before selecting.

Selecting Single Entities

Whenever you want to select entities one at a time (even if you want to pick several of them) do the following:

1. Move the cursor through the screen. FEMAP will highlight different entities as you move the mouse over the screen.
2. Click the left mouse button when the entity you want is highlighted. This action places the entity ID directly into the selection list.
3. If you made a mistake by picking the wrong entity, you can either use the *Delete* button to remove it, or change to *Remove/Exclude* mode and pick it again.
4. Repeat the previous step until all entities have been selected (or use any of the other selection methods), then press *OK* to complete the selection. You will notice that once an entity has been chosen, it is no longer dynamically highlighted, so you may more easily choose from the remaining entities on the screen.

Or, alternatively:

1. Move the cursor to point at the entity and double click the left mouse button. This places the entity directly into the selection list and presses *OK*. No further input is required, but you will not be able to correct any mistakes.
2. You can use this technique in combination with the previous “single click” method by just “double-clicking” the last entity that you want to select.

Remember, by changing the *Add, Remove, Exclude* setting, you can either select or deselect entities.

When you are selecting single entities, the entity that is selected is based on where you point in the Graphics window, and what you have previously selected. Any entity that is already in the selection list will be skipped as FEMAP looks for the entity closest to your selection. This means that you can pick three times at the same location to choose the three entities which are closest to that location. FEMAP will not pick the same entity three times.

Selecting Multiple Entities

One of the most powerful graphical selection capabilities of FEMAP is the use of the pick method described in the *Entity Selection* dialog box explanation above. You have access to *Box, Circle, Polygon*, and *Freehand* picking. Each of these methods were explained above and will not be reiterated here. The *Box, Circle,* and *Polygon* picking are unique, however, in that they can be accessed while in normal mode. By pressing the *Shift* or *Ctrl* keys (or both at once), you can select all entities which lie inside a desired area. The *Shift* key enables you to define a box, while the *Ctrl* key will allow you to define a circular area. Holding both Shift and Ctrl at once, will allow you to define an area in the shape of a polygon. To perform graphical selection, simply follow the steps below:

1. Press and hold down the *Shift* key if you want to select inside a rectangular area, the *Ctrl* key if you want a circular area, or both the *Shift* and *Ctrl* keys at the same time if you want to begin creating a polygon.
2. Point at one corner of the rectangular region (or the center of the circular region). For polygon picking, choose the location of the first point of the polygon.
3. Press and hold the left mouse button. For polygon, click the left mouse button to begin choosing the points of the polygon.
4. Move the cursor. As you move, you will see a box/circle which represents the area that you are selecting. When the box/circle surrounds the area that you want, release the left mouse button. This will select all entities inside the area and add them to the selection list. You do not have to press *More*. For polygon picking, choose any number of points to define an area in the shape of any polygon.

5. Make additional selections, or click the *OK* button when you have selected all of the desired entities.

6. To abort a selection of this type, just release the *Shift* or *Ctrl* key prior to releasing the left mouse button. No selection will be made.

The *Freehand* picking method works almost identically to the circle and box picking except it actually traces the history of your movement (as opposed to just using the two end points). Polygon picking is just slightly different in that it is not based upon dragging the cursor, but rather you must select each individual location of the polygon.

The following tips will help you get started with graphical and multiple entity selection.

1. If you need to select many entities in a complex region, you can combine the area selection techniques with the *Add, Remove, Exclude* options. By choosing add, you can combine multiple overlapping square, circular, polygon, and freehand regions. By choosing remove or exclude, you can subtract additional selections, effectively cutting holes in your selection region.

2. Since entity selection is used by so many commands, you may find yourself wanting to select the same entities over and over again for multiple commands. If you just want the same selection for a few commands, the *Previous* button will recall your selections. If you need to come back to this selection sometime later, it is best to use the *Group* options to define those entities as a group. Every time you need them, you can simply use the *Group* drop-down list to retrieve the selection no matter how complicated it might have been. Remember to give the group a title so you can remember which one to pick!

3. If you are working with a complex model, cursor selection can take a while both for you and for the computer to determine which entity is closest to your pick. If you define a part of your model as a group, and then only display that group (use the *Group* tab of the *View, Visibility* command or the “visibility check boxes” for groups in the *Model Info* tree), the process can be much simpler.

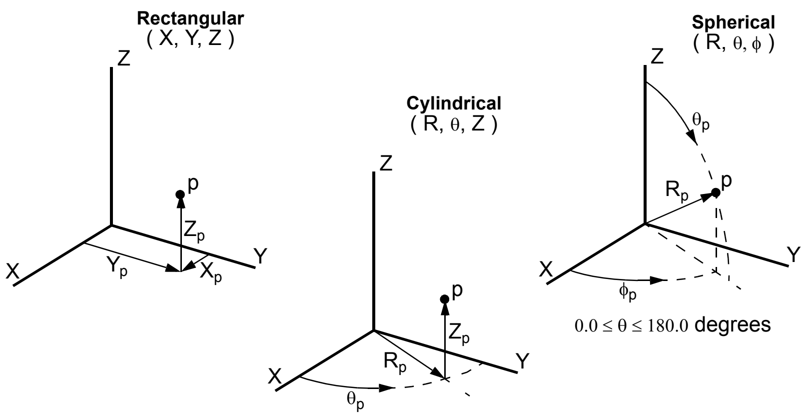
4. The cursor snap mode is used for all cursor selections including selection of entities. If you are snapping to a grid, node or point, you must remember that the entity to be selected will be the one closest to the grid, node or point that was “snapped-to”, not necessarily the one closest to the location you picked. The same principle applies to area picking. The corners of the area are changed by the snapping action.

4.3.2 Coordinate Definition

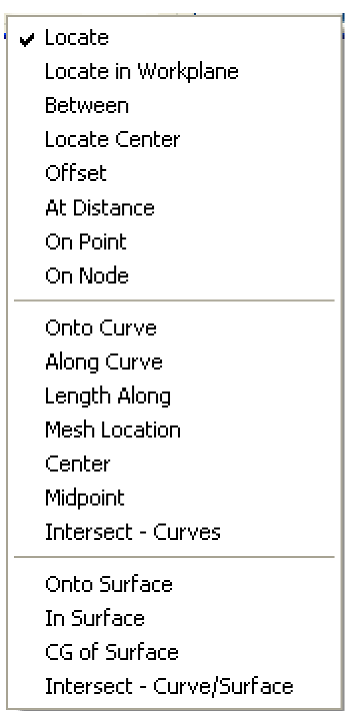
One of the most common actions in FEMAP is to define the coordinates of a desired location. In the most basic form this simply involves specifying the three-dimensional (X, Y, Z) coordinates of the location in the global rectangular coordinate system. In addition, coordinates can be specified in global cylindrical, global spherical or relative to any other coordinate system you create.

Note: Changing the snap mode to *Smart Snap*, *Snap to Point*, or *Snap to Node* can aid in selection of coordinates by using entities which already exist in the model. *Smart Snap*, in particular, can help minimize how often the *Method* needs to be changed by allowing "smart snapping" to any node, any point, the midpoint of any curve, or the center point of any circular arc.

The following figure shows the conventions for entering coordinates in any type of FEMAP coordinate system. The conventions shown in this figure are used throughout FEMAP. Whether you are actually specifying a coordinate, defining a vector, defining a plane or entering some other coordinate related data, these conventions are your key to interpreting the input which is required.



Note: Throughout FEMAP, all angular dimensions must be specified in degrees.



In many cases, you cannot easily determine coordinates. For these times, FEMAP provides numerous alternative coordinate definition methods which allow you to specify the coordinates in terms of quantities or entities that you do know. With any of the methods, you can use any of the global or user defined coordinate systems to further simplify your input.

All of the coordinate definition methods provide a *Method* button which allows you to switch to another coordinate definition method. Switching methods involves selecting an option from the popup menu.

When you start a model, some of the methods will be unavailable. For example, you cannot use *On Node* if you do not have any nodes. All of the methods will automatically become available as soon as the required entities are created.

FEMAP is a full three-dimensional modeling program. All coordinates are always specified with three coordinates, relative to one of the global or user-defined coordinate systems. The FEMAP workplane is only used for graphical selections and to orient geometry created by certain geometry creation commands.

The *Locate* method is the default when you start FEMAP and the *Locate* coordinate definition dialog will be displayed by every command that requires coordinates. If you switch to a different method, that method will become the default for all commands until you switch again.

Features for All Methods

Located near the bottom and right of all of the coordinate definition dialog boxes are several common controls.

ID:

Indicates the ID of a point or node to be created. If you are not creating a point or node, this field will be disabled. The ID will automatically increment after each creation, or you can enter the ID of any point or node which does not already exist.

CSys:

Specifies the definition coordinate system in which you will enter the X,Y,Z location (or other method). The drop-down list will contain all of the available coordinate systems for your choice, or you can select a coordinate system from any graphics window using the cursor. Changing the definition coordinate system will automatically transform any coordinates that you have already entered into the new system. The X,Y,Z titles will also change, based on the type of the active definition coordinate system. For cylindrical systems, XYZ will become RTZ (R, Theta, Z). For spherical systems, XYZ will become RTP (R, Theta, Phi).

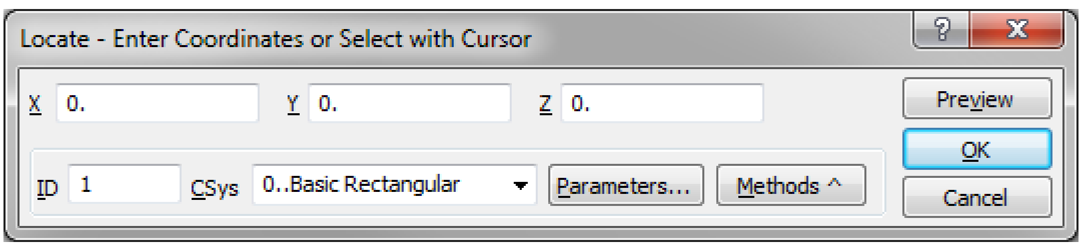
Parameters:

This is another option that is only available when you are creating points or nodes. It allows you to specify additional parameters for those entities. For more information, see Section 3.1.1, "Geometry, Point..." and Section 4.2.1, "Model, Node..." in *FEMAP Commands*.

Preview:

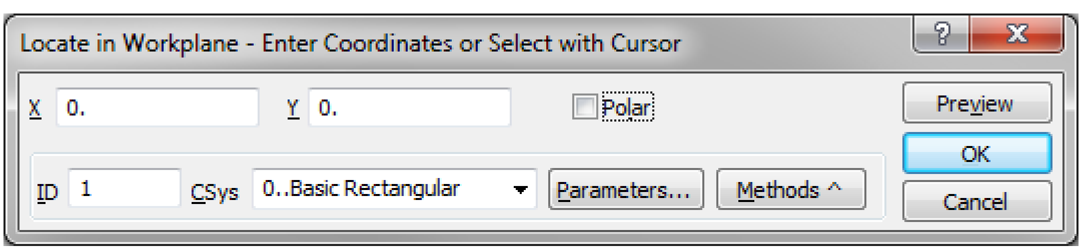
Draws a dot in the graphics windows at the location currently being defined. You can use this option to see where the coordinate will be prior to choosing *OK* to accept the value. Choosing *Preview* after you select coordinates with the cursor does not provide any new information. Cursor selection automatically shows the location being picked. If you type input, or modify a cursor selection however, *Preview* will show you the location.

Coordinate Locate Method



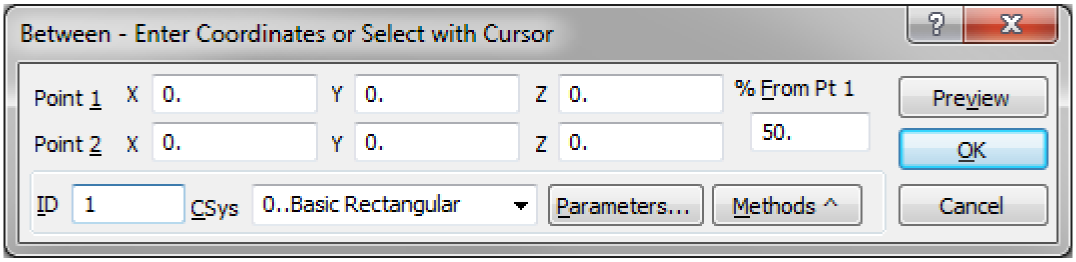
This method allows you to directly specify a location. As always, coordinates are relative to the definition coordinate system. When using this method, you are simply specifying the coordinates directly, as shown in the previous coordinate definition conventions picture. Remember however, that the various cursor snapping modes can be use to adjust the coordinates that you choose graphically.

Coordinate Locate In Workplane Method

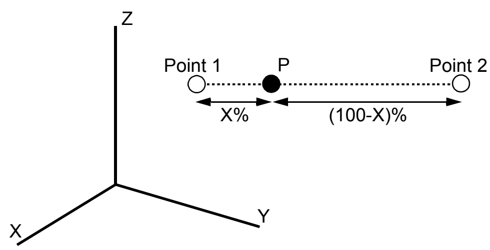


This method is very similar to the *Locate* method, except only two coordinates are required, X and Y in the workplane.

Coordinate Between Method

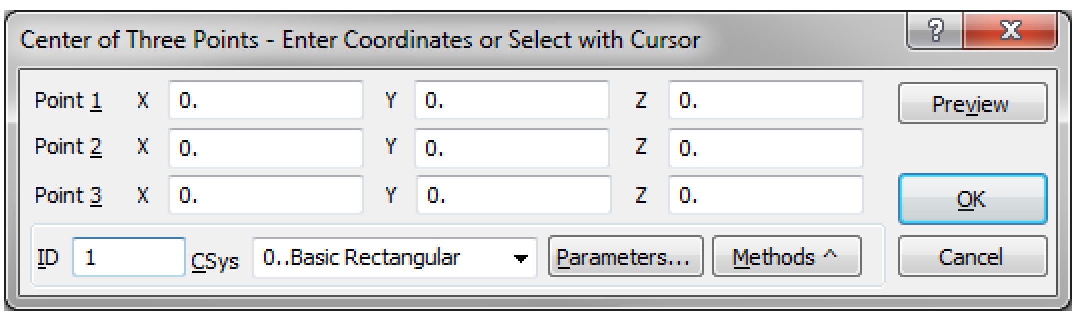


The *Between* method allows you to interpolate between two other locations. In addition to the two endpoints, the coordinates are determined from a percent of the distance from the first location to the second location. Just as the endpoint locations are specified in the definition coordinate system, the interpolation is also done in that coordinate system. If the definition coordinate system is non-rectangular, the resulting point may not lie along a straight line between the endpoints. For example, in a cylindrical system (R, Theta, Z), a location 50% of the way between the endpoints (1,0,0) and (1,90,0) is (1,45,0). The interpolation was carried out along the cylindrical arc.

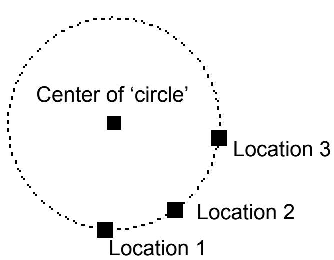


Hint: Use this method to locate coordinates based on the positions of two existing nodes or points. Set your cursor snap mode to *Node* or *Point* and select the endpoints with the cursor. Complete your selection by typing the desired percentage from the first endpoint.

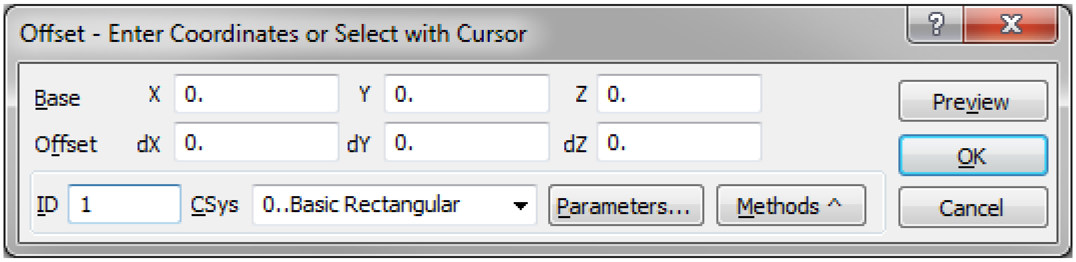
Coordinate Locate Center Method



The *Locate Center* method requires three specified locations which are not colinear to determine a “circle”. The “center” location is then determined by finding the center point of the “circle”. A geometric circular curve is NOT created.

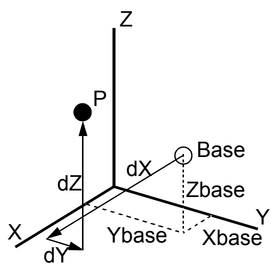


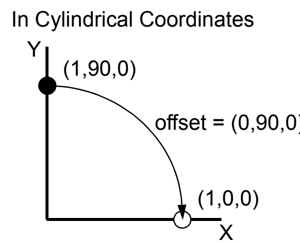
Coordinate Offset Method



Offset coordinates are a variation of the *Locate* method. You must specify a *Base* location (just like *Locate*), but in addition, you can specify an offset from that location. The offsets are delta coordinates which are added to the base location, they are not a vector. In rectangular coordinates this distinction does not make any difference. In cylindrical or spherical coordinates however it can change the resulting location. For example in cylindrical coordinates (R, Theta, Z), if the base is (1,0,0) and the offset is (0,90,0), then the resulting location is (1,90,0), which is not in the Theta tangent vector direction from (1,0,0)

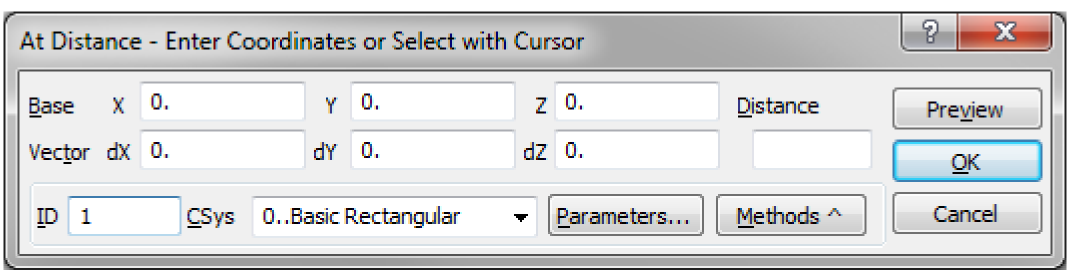
.



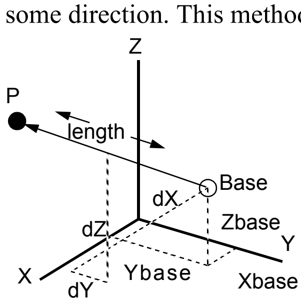


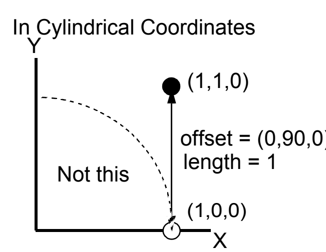
Use this method if you want to specify coordinates which are offset from a node or point. Set the base location by picking the desired node or point (with the cursor snap mode set to *Node* or *Point*). Then just type the desired offset

Coordinate At Distance Method



This method is similar to the offset method. You still specify a base. Instead of an absolute offset however, these coordinates are defined by a vector direction and a distance. This approach is useful when you want to offset a specific distance along some direction. This method does not use delta coordinates. It always offsets along the vector.





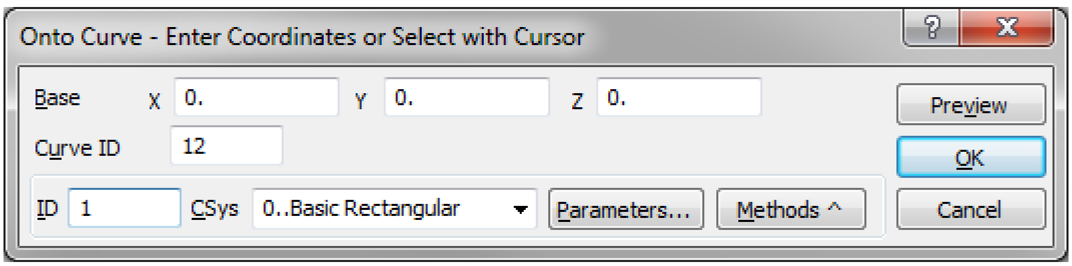
Coordinate On Point Method

This method defines coordinates which are identical to the location of the selected point and requires input of the Point ID only. *On Point* is disabled unless you have at least one existing point. If you set the cursor to snap to the nearest point, you can specify the same coordinates as *On Point* using the *Locat*e method. Be careful if you are using this method to create new points or nodes. They will be coincident with the point you select, and difficult to see.

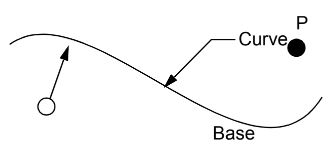
Coordinate On Node Method

This is identical to the *On Point* method except that the coordinates are chosen at the location of a selected node.

Coordinate Onto Curve Method

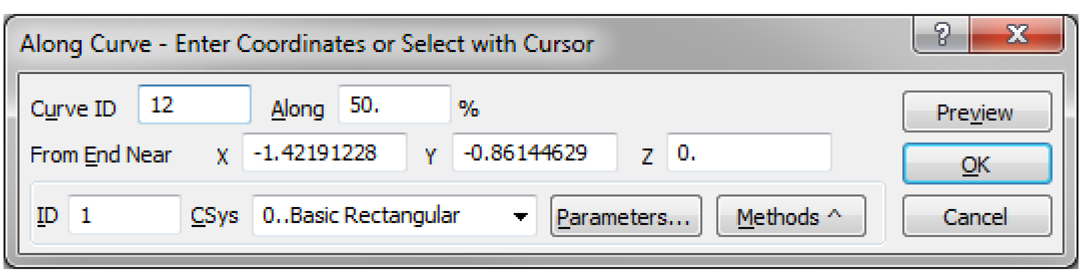


The *Coordinate Onto Curve* method projects a location onto a curve. The direction of the projection is always perpendicular to the curve. For example if you are projecting onto an arc or circle, the specified coordinates are first projected onto the plane of the curve and then toward (or away from) the center of the curve, to a location on the perimeter.

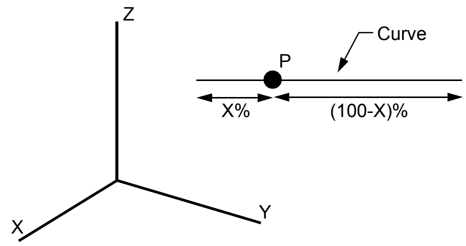


Note: Remember, all curves are considered infinite. If you choose a base location past the end of a line segment, it will be projected onto the extended line, not to the endpoint of the segment.

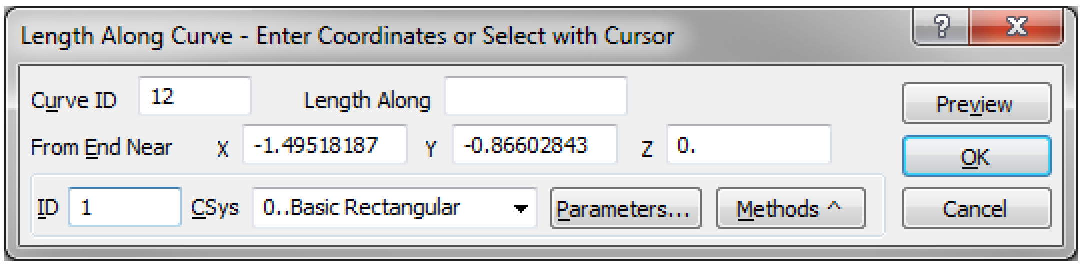
Coordinate Along Curve Method



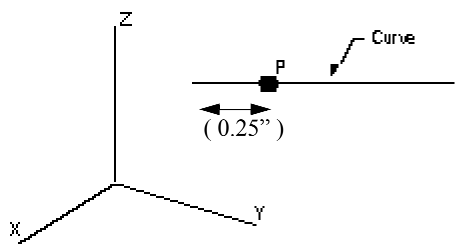
The *Along Curve* method allows you to select coordinates along a curve. You must identify the curve and a percentage along the length of the curve. The location is calculated using the percentage of the curve length from the end of the curve which is closest to the *End Near* location. This is a quick method to define a location at any position along a curve.



Coordinate Length Along Method

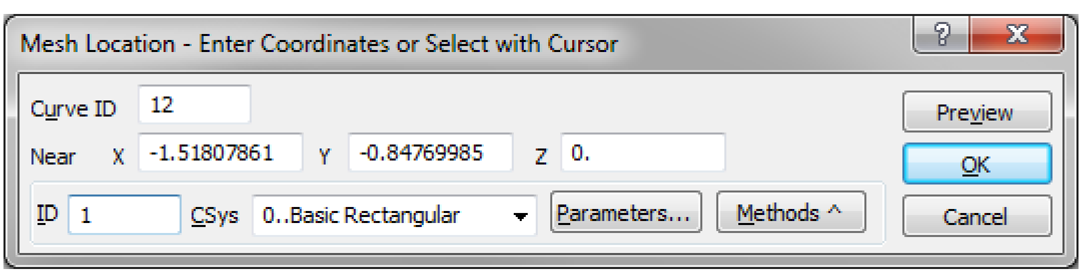


The *Length Along* method allows you to select coordinates at a distance from one end of a curve. You must select the curve and the distance along the curve. The location is determined by moving along the curve the *Length Along* value from the end of the curve closest to the *End Near* Location.

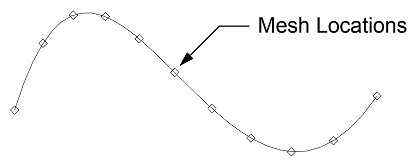


Note: If you select the curve with the mouse, the *End Near* location will be automatically updated to the point where you made your selection. By selecting the curve near the end that you want to measure from, you can automatically specify *End Near* with no further input.

Coordinate Mesh Location Method



The *Mesh Location* method selects coordinates based on the mesh size which you have defined for a curve or its points. If no mesh size is defined for the selected curve, the mesh size will be determined from the mesh size defined for the curve points or the default mesh size. In addition to the curve, you must specify a location near to the mesh location that you want to select. FEMAP first selects the curve, and then finds the closest mesh location to the coordinates that you specified.



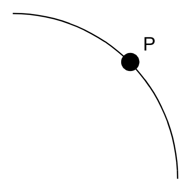
Near is automatically defined as the location you pick if you select the curve graphically. You do not have to specify any additional input.

Coordinate Center Method

This method is a quick way to select the center of an arc or circle. Simply identify the arc or circle you want to use. You cannot choose any other type of curve for this method. Refer to the *Midpoint* method for other curve types. As an alternative for arcs and circles you can use the *On Point* method, since the center of the arc or circle is always defined by a point.

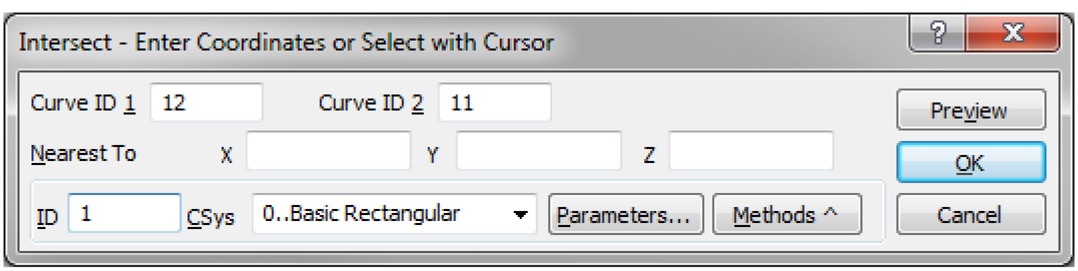
Coordinate Midpoint Method

The *Coordinate Midpoint* method is a simple way to select coordinates in the middle of a curve. These coordinates always lie along the curve. For example, they lie on the perimeter of an arc, at an equal arc length from the beginning and end of the arc. For a line, the point is simply half way between the endpoints.

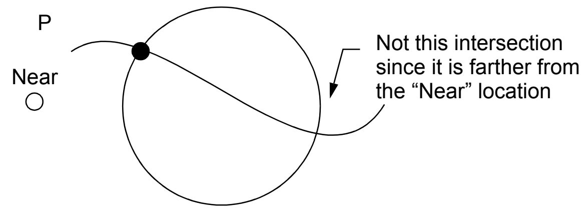


The only input required for this method is to select the curve that you want to use.

Coordinate Intersect - Curves Method



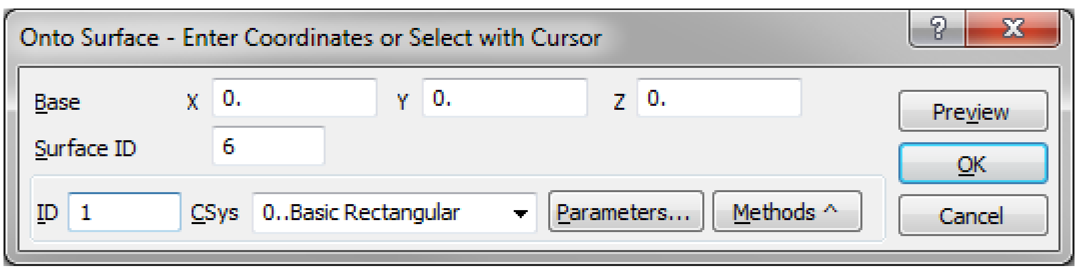
The *Coordinate Intersect* method defines coordinates at the intersection of two curves. You must select the curves that you want to intersect. In addition, you must specify a location near the intersection. In fact, this location is not required if you are intersecting lines since there is only one possible intersection location. For other curves however, where multiple intersection locations can exist, the intersection which is closest to the coordinates that you specify is computed.



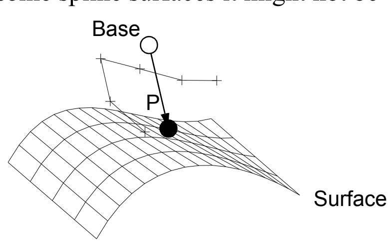
This method considers all curves as infinite. That is, lines are extended in both directions to infinity and arcs are extended into circles. The intersection location does not have to fall between the endpoints of the original curves.

Hint: The *Nearest To* location is automatically updated if you select the second curve graphically. By selecting the curve near the point of intersection, you will not have to specify any further input.

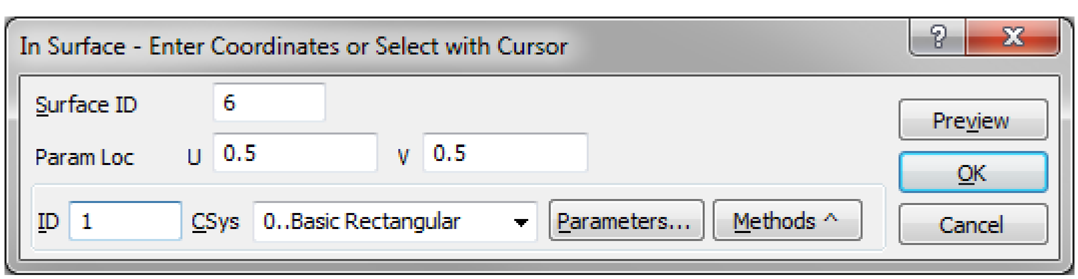
Coordinate Onto Surface Method



The *Coordinate Onto Surface* method is similar to *Onto Curve*. It projects the base location onto a surface. In this case the projection is toward the point on the surface which is closest to the original. Typically this direction is perpendicular to the surface, but for some spline surfaces it might not be



Coordinate In Surface Method

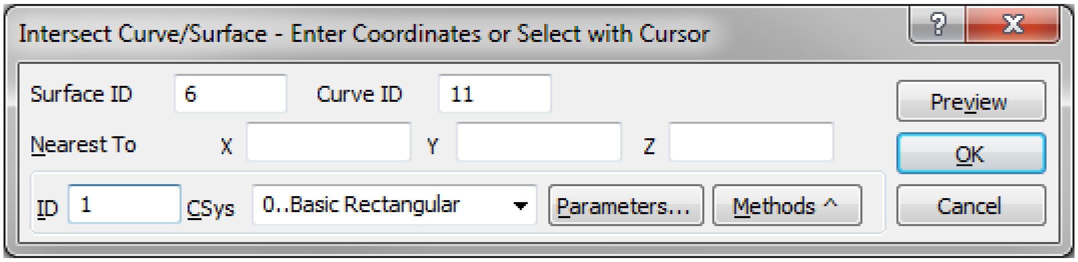


The *Coordinate In Surface* allows you to define a location based upon a parametric location on a surface. The only input required for this command is the surface ID and the u,v location. The values for u,v must be between 0 and 1.

Coordinate GC of Surface

The *Coordinate - CG of Surface* allows you to define a location at the center of gravity of a selected surface.

Coordinate Intersect Curve/Surface



The *Coordinate Intersect Curve/Surface* option allows you to define a location based upon the intersection of a solid model surface (Parasolid) and a curve. This option cannot be used if you do not have Parasolid surfaces in your model and will be grayed. Neither boundary surfaces or FEMAP standard surfaces can be used with this command.

Simply select the surface and curve, and a location near the intersection (in case of multiple intersection points) and FEMAP will compute the location of intersection

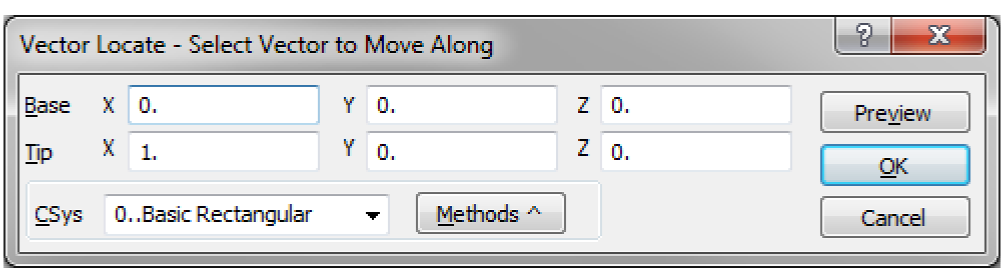
4.3.3 Vector Definition

Just as there are many methods to enter coordinates, there are many methods for defining a vector in FEMAP. Some vectors in FEMAP are just used for determining a direction. They do not require a length. The axis around which you rotate a group of nodes is an example of this type of vector. Other vectors not only require a direction, but also a length. The vector which you translate nodes along in the *Move By* command is an example of this vector type. For either vector type, all methods are available. Some methods, like *Locate*, implicitly define their length based on the normal vector input. Other methods, like *Axis*, require you to define an explicit length whenever the vector requires a length.

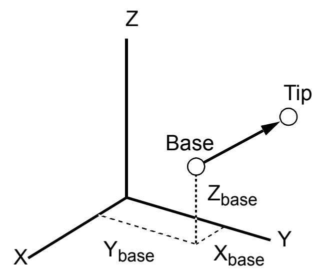
All coordinates and vector components required for various vector definition methods must be input in the active definition coordinate system. A drop-down list (CSys) is available in each of the dialog boxes to choose the coordinate system. In addition, when you change coordinate systems (or methods), current entries are transformed to an equivalent vector in the new system. Therefore, you can enter part of the data using one coordinate system or method, and then switch to a new coordinate system or method to complete the definition.

Just like coordinates, you can use the cursor to define the vector. For methods that let you define the vector tip, in addition to the graphics cursor, you will see a vector coming from the base location and attached to the cursor. For more complex methods, Bisect and Normal, additional construction lines are visible. To see the vector prior to accepting the input, pick the *Preview* button. This will draw the vector in all graphics windows.

Vector Locate Method



This method defines a vector which goes from a base coordinate to a tip coordinate. The vector length, if required, is the distance between the two coordinates. As always, all input is in the active definition coordinate system.

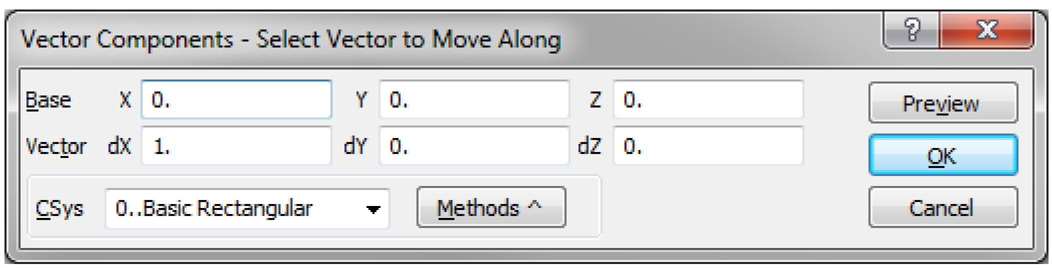


Use this method when you know two existing points that the vector should go between.

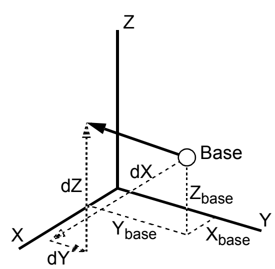
Vector Locate/Length Method

This method is very similar to *Vector Locate*, as described above. You still specify two points, but you also specify a length. This specified length is used instead of the distance between the two points.

Vector Components Method

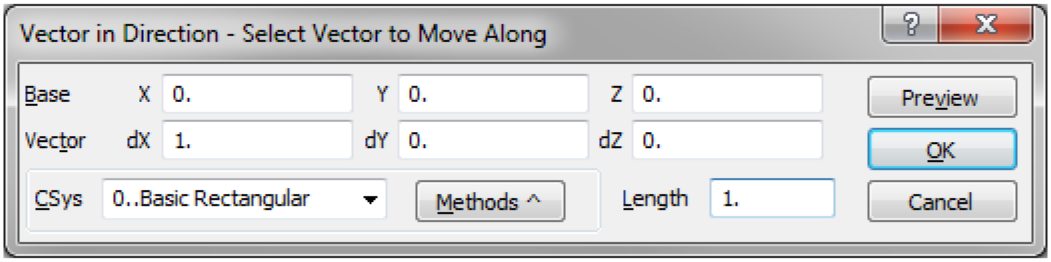


This method creates a vector by specifying a base location and the components of a vector. The vector length is determined by the magnitude of the components that you specify. Use this method when you want to specify a vector or direction with specific offsets from a base location.

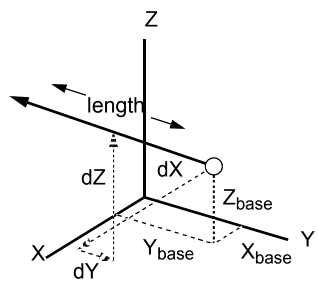


Note: When using a non-rectangular coordinate system, vector components are measured along principal directions at the base location. For example, if in global cylindrical coordinates, you specify a base of (1,45,0), and vector components of (0,90,0), this implies a vector of 90 inches (length units) in the positive theta direction at (1,45,0), or 135 degrees from the global X axis. It does not imply a change in theta of 90 degrees.

Vector Direction Method



This method is identical to the *Vector Component* method if you are defining a direction vector (one with no length). If length is required, this method allows you to specify it explicitly. It is not determined from the delta coordinates. Use this method when you want to specify a vector in a certain direction of a specific length.



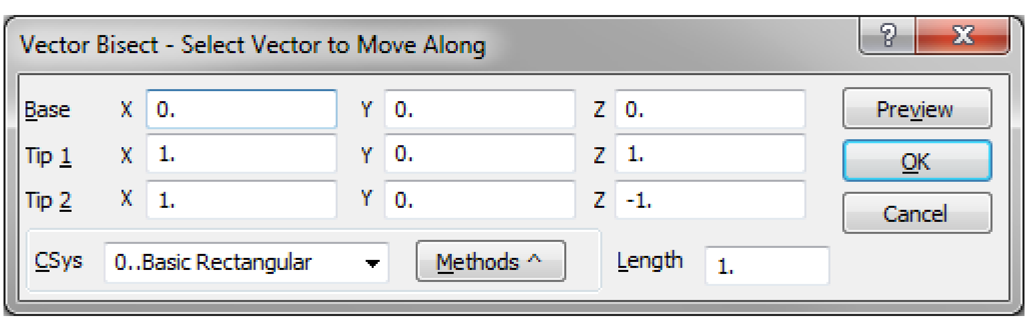
Vector Points Method

This method is identical to *Vector Locate*, except that the vector lies between two existing points. You can mimic this method using *Locate* by setting the cursor to snap to points and selecting the same two points. To use this method, simply select the two points (you must have at least two points in your model to use this method).

Vector Nodes Method

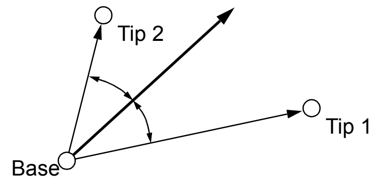
Again, this method is identical to *Vector Locate*, except that the vector lies between two existing nodes. You can mimic this method using *Locate* by setting the cursor to snap to nodes and selecting the same two nodes. To use this method, you must have at least two nodes in your model.

Vector Bisect Method



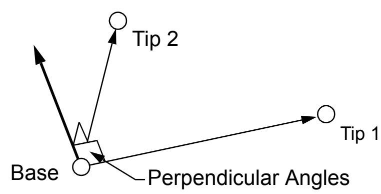
This method will define a vector which bisects two other vectors. The two “construction” vectors are defined by a common base location and the location of their respective tips. The bisecting vector always lies in the plane formed by the three points, which must not be colinear. You must explicitly define the length of the bisecting vector if it is required. It is not determined from the lengths of the “construction” vectors

.

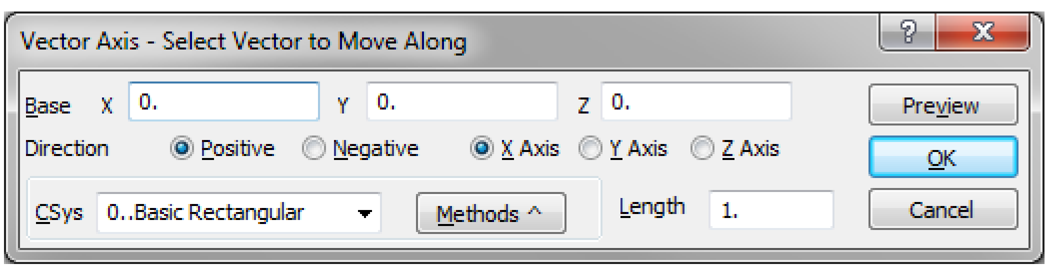


Vector Normal Method

This method is similar to *Vector Bisect* and requires the exact same input. Instead of bisecting the “construction” vectors however, this is oriented normal to the plane formed by the “construction” vectors. It is still located at the base location. The positive vector direction is determined by the “right-hand rule” from the first “construction” vector toward the second. Again when it is required, you must explicitly define the length. It is not determined from the “construction” vectors.



Vector Axis Method



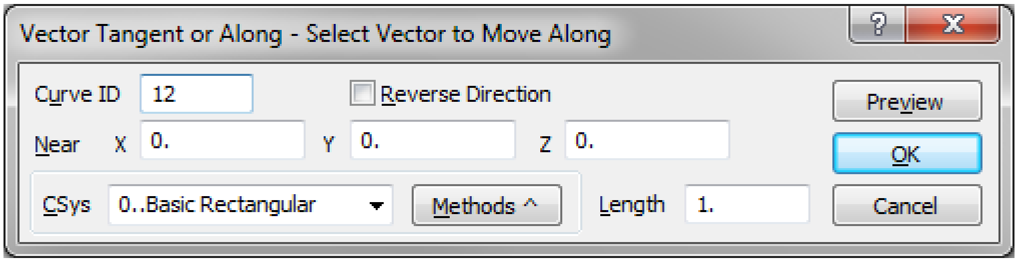
This method is unlike all preceding methods in that the only coordinates you specify are for the base point. The direction of the vector defined by this method is based on one of the positive or negative axis directions of the active definition coordinate system. When required, the length must be specified explicitly. If you have already defined coordinate systems in the desired direction(s), this is one of the easiest and quickest methods to define a vector.

If the active coordinate system is non-rectangular, the axis locations refer to the coordinate directions at the base point. For example, in a cylindrical coordinate system (R, Theta, Z), the Y axis refers to the Theta direction at the selected base point.

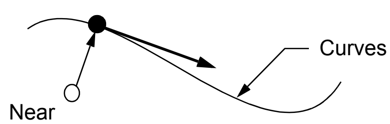
Vector Global Axis Method

This method is much like *Vector Axis*, except that the vector is always in one of the axis directions of the global rectangular coordinate system. The definition coordinate system is only used for convenience in entering the base point. It has no effect on the vector direction. For this reason, it does not matter whether it is rectangular, cylindrical or spherical. Again, with this method, you must explicitly define the length whenever it is required.

Vector Tangent or Along a Curve Method

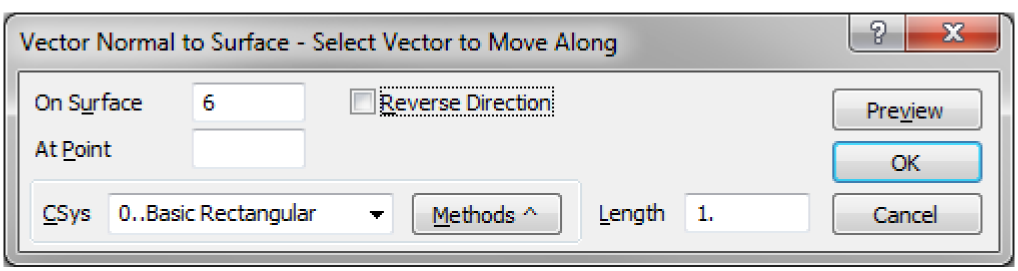


This method allows you to create a vector which is tangent to a curve. If you choose a line, the vector will be along the length of the line. In addition to the curve, you must choose a location. This location is projected onto the curve, and serves as the base for the vector. The vector direction is determined automatically from the tangent to the curve at the projected location.



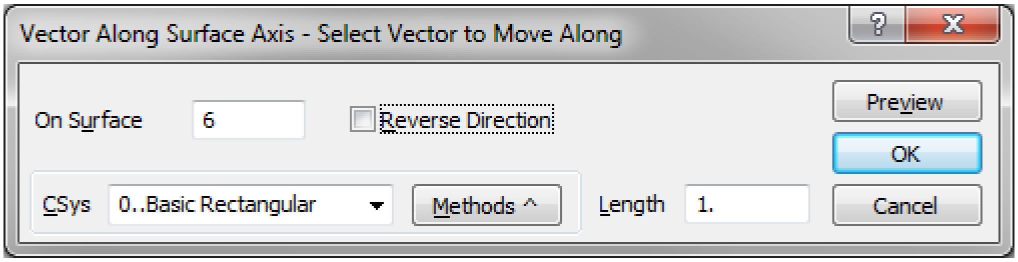
Normally, tangent vector always goes from the start (first end) of the curve toward the end of the curve. If you check *Reverse Direction* the tangent will go in the opposite direction. If you are unsure of how the curve was created, press *Preview*. Then, if the vector is pointing in the wrong direction, reverse the current direction by clicking *Reverse Direction*. If you use this method to specify a vector that requires a length, you must explicitly define the length since no length is implied by the tangent direction.

Vector Normal To Surface



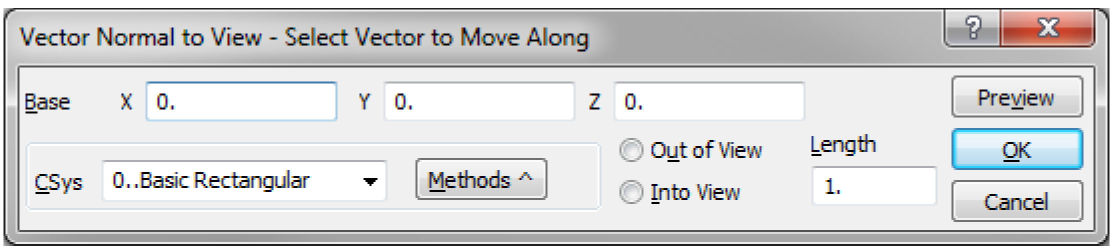
This method defines a vector which is normal to a surface at a particular location. The input for this method is simply the surface, the point, and length (if required). You may also choose to reverse the direction of the vector so it points in the negative normal direction.

Vector Axis of Revolution



This method defines a vector which is the axis of revolution for a revolved surface. The input for this method is simply the surface and a length (optional). You may also choose to reverse the direction of the vector so it points in the negative direction of the surface's axis of revolution.

Vector Normal To View Method



This method defines a vector which is normal to the active Graphics window. If there are no graphics windows, it defines a vector parallel to global Z. The direction of the vector is either into the view or out of the view (screen), depending upon the option chosen. When required, the length must be explicitly specified.

This method is often very useful in combination with the various *View Align* and *View Rotate* commands to specify vectors in skewed directions. You can first align the view correctly, see that everything is correct, and then easily choose the vector with a minimum amount of input without worrying about the direction.

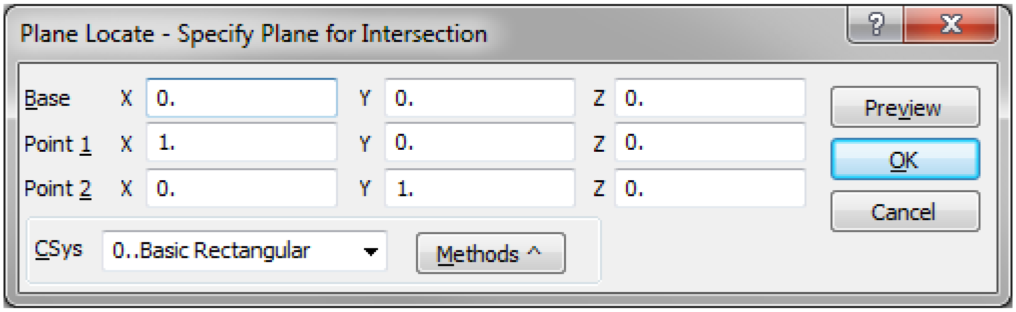
4.3.4 Plane Definition

FEMAP also provides multiple methods to define planes. All definition methods create a plane which passes through an origin location and which is oriented by a vector normal to the plane. In most cases, like specifying a plane to reflect about, coordinate directions in the plane are not required. When they are required however, (for example, when you define the workplane), "in-plane" coordinate directions are automatically determined based on your existing input.

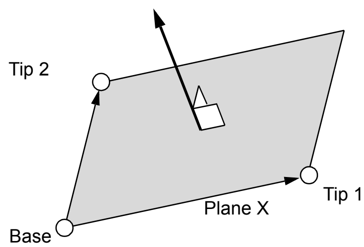
When you use the cursor to define a plane you will see several additions to the graphics cursor. As you specify the first vector used to define the plane, you will see a vector attached to the cursor. Then as you specify the final vector/direction, the plane will be dynamically shown on the screen. Just as in vector definitions, the more complex methods, *Bisect* and *Normal*, will also draw additional construction lines. If you want to see the final plane prior to accepting your input, whether you used the keyboard or mouse, pick the *Preview* button. This will draw the plane in all graphics windows.

Note: Unlike vector definition, it is often necessary to press *Preview* to see an accurate orientation of the plane - even if you use the cursor to define the plane. This is especially true if you are using cursor snapping. Small movements of definition locations due to snapping can make large changes in plane orientation.

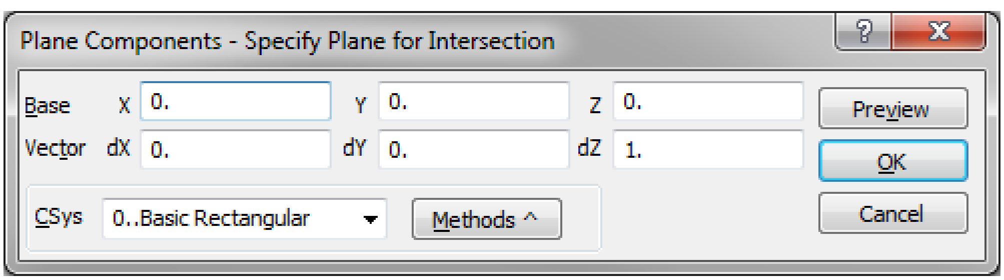
Plane Locate Method



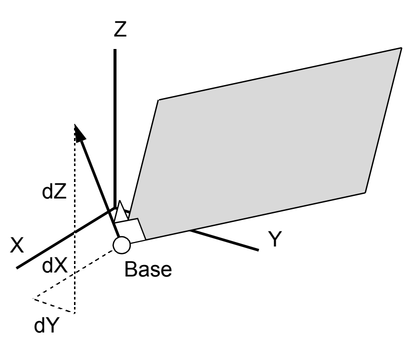
This method is the default plane definition method. It involves specifying three, non-colinear locations which define the plane, a base (origin) and two other locations. The plane normal is determined from the cross-product of the vector from the plane origin to the first location and the vector to the second location. The vector from the origin to the first location also defines the in-plane X direction. All input is in the definition coordinate system.



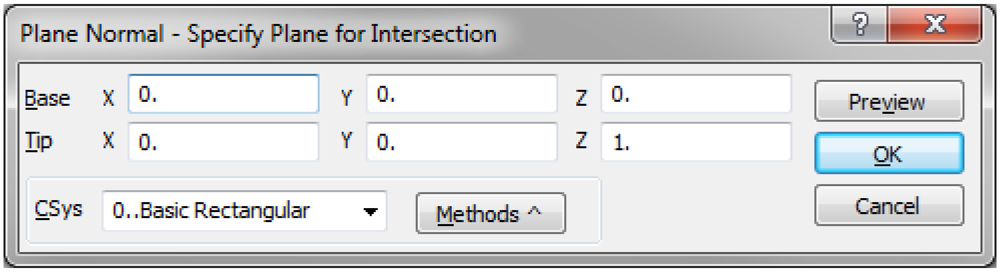
Plane Components Method



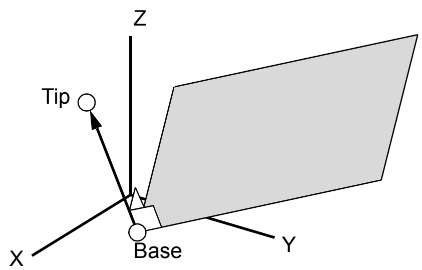
Defining a plane by components requires specifying an origin and the components of a vector which is normal to the plane. The local X direction in the plane is automatically determined by calculating the cross product of the global Y axis and the plane normal. If the plane normal lies along the global Y axis, then the local X direction is set to lie along the global X axis.



Plane Normal Method



The plane normal method is similar to the *Plane Components* method. In this case however, you must specify the base/origin and a point at the tip of the normal vector (as opposed to the components of the normal vector). The in-plane X direction is determined in the same manner as for the *Plane Components* method.



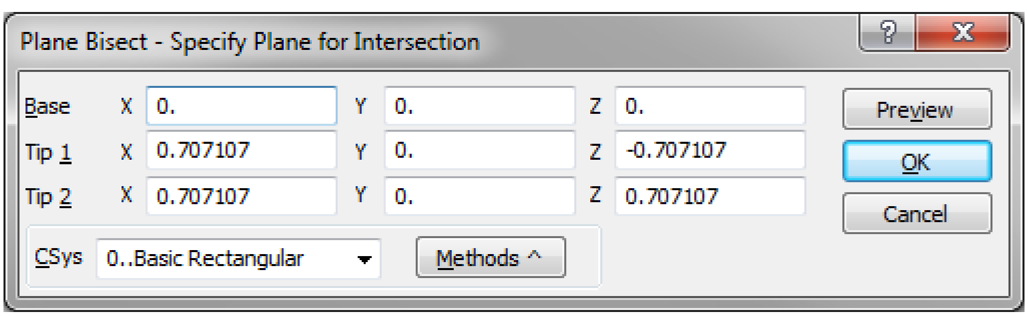
Plane Points Method

Plane Points is just like the *Plane Locate* command except that the locations are specified using existing Points.

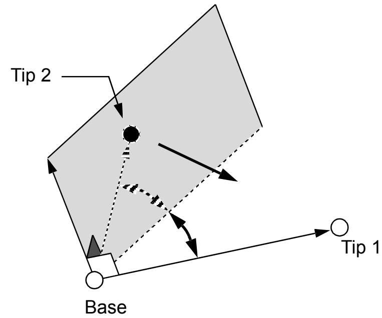
Plane Nodes Method

Plane Nodes is just like the *Plane Locate* command except that the locations are specified using existing nodes.

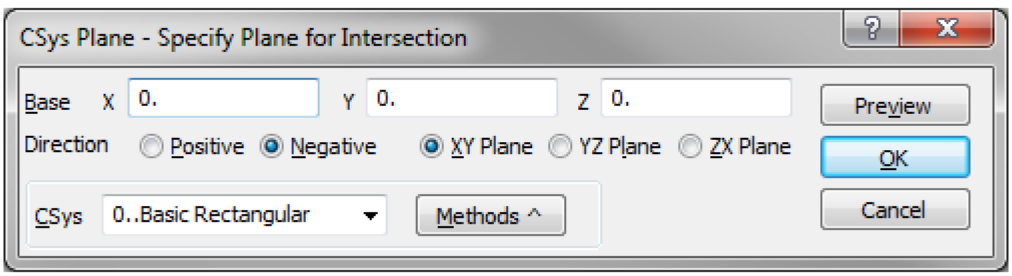
Plane Bisect Method



The *Plane Bisect* method is similar to the *Vector Bisect* method of specifying a vector. It requires specifying a base and two other vector tip locations. The resulting plane bisects those two vectors. It is normal to the plane formed by the two vectors and oriented such that it lies midway between the vectors, through the plane base/origin. The normal to the plane is in the plane formed by the construction vectors, and points toward the first vector. The in-plane X direction is defined in the plane of the construction vectors.



Plane Csys Plane Method



This method simply chooses one of the principal planes (XY, YZ, or ZX) of the definition coordinate system. The normal can face in either the positive or negative direction. The in-plane X direction is determined by the first letter in the plane definition. That is, the X direction for an XY plane is along the X axis, the X direction for a YZ plane is along the Y axis and for a ZX plane, along the Z axis.

If you choose a plane in a non-rectangular coordinate system, the plane normal is defined by the direction of the coordinate tangent at the base/origin location. For example, in a cylindrical coordinate system, with the origin set to (1,45,0), a ZX plane is rotated 45 degrees from where it would be if the coordinate system were rectangular. This method is very convenient if you already have a coordinate system defined that is properly aligned to the directions you need to select.

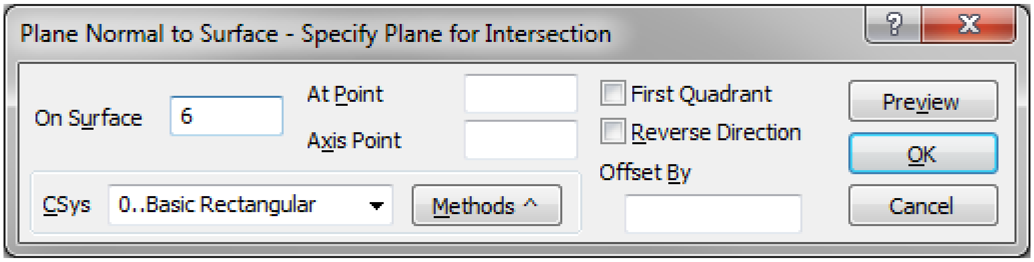
Plane Global Plane Method

The *Global Plane* method is identical to the *CSys Plane* method, except that it always chooses a plane aligned with the principal directions of the global coordinate system instead of the selected definition coordinate system. Since the global system is rectangular, the special cases for non-rectangular coordinate systems do not apply to this method. This is the easiest method to align a plane with the global axes.

Plane Align to Curve Method

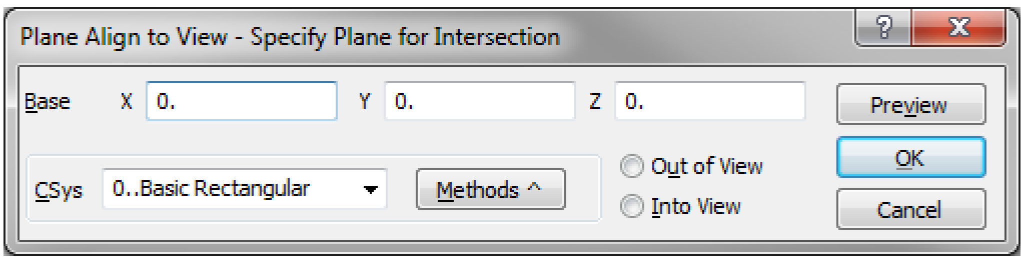
This method allows you to quickly move the workplane, or set any other plane to the plane of an arc or circle. Other types of curves cannot be used. The workplane origin will be moved to the center of the arc or circle that you choose. The workplane normal will be along the normal to the curve and the workplane X direction will be toward the first point on the curve boundary. The only input required for this method is the curve ID.

Plane Surface Normal Method



This method allows you to quickly align the workplane, or set any other plane to a specific surface. The only input required for this method is the Surface ID and the point of the origin (*At Point*). You may also specify an axis point to align the X axis of the plane. Other options include an *Offset Value, Reverse Direction of the Plane Normal,* and force the first quadrant of the plane to contain the surface (this may flip the plane normal as well).

Plane Align to View Method



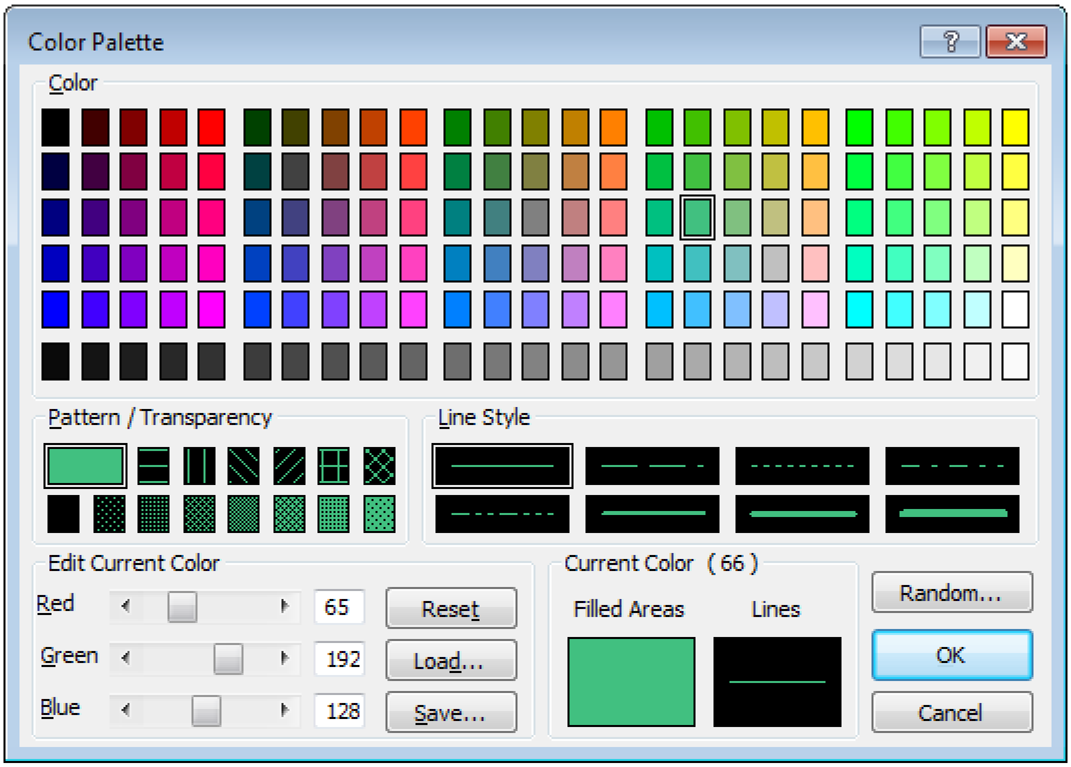
This method is just like the *Vector* definition method *Normal to View*. The resulting plane will pass through the specified base/origin and will be parallel to the plane of the screen. The normal direction can be specified as either into or out of the view. The in-plane X direction is aligned with the *View X* (horizontal) direction.

4.3.5 Color Palette

Throughout FEMAP you will see many dialog boxes with one or more text boxes which allow you to choose a color. If you know the color ID, you can just enter it into the text box. To the right of these text boxes, however, is a command button titled *Palette*. Choosing the *Palette* button will display the standard FEMAP *Color Palette* from which you can select the color graphically. After selecting the color, choose *OK*, and the text box will automatically be filled with the ID of the color you selected.

The color palette consists of 125 colors which, by default, span the entire color spectrum, plus an additional 25 Gray-scale colors. Some graphics adapters (like the standard VGA) cannot display this many colors simultaneously. For those adapters, the additional colors will be displayed as dithered patterns for any filled areas.

The Color Palette is saved with the model file.



For filled areas, you can also choose one of the available hatched patterns instead of the solid fill. Hatched patterns use the line color, not the dithered fill color. In addition, the second row of patterns are transparent colors. You will be able to “see through” areas that are filled with one of these patterns. Eight different transparency levels are provided by the eight patterns. These range from completely transparent to nearly opaque. The partially transparent colors will combine with colors from any other geometry and will overpaint to produce a tint. These transparent colors however use the “solid” line colors. You will therefore get the best results if your graphics adapter can display 256 or more colors.

Note: The eight “transparency” patterns apply to all entities which can have color (i.e., nodes, elements, points, lines, surfaces, regions, etc.).

For lines, you can select a style which is either patterned (long and short dashes) or thick. If you choose one of the patterned linestyles, it may look solid if you are drawing very short line segments. This can often happen with arcs, circles and splines if you set the *Curve Accuracy* (in *View Options*) very small. Since FEMAP approximates these curves with straight lines, setting a very small accuracy results in many very short line segments. To see patterning on these curves you will have to increase the *Curve Accuracy* value, resulting in fewer line segments and less precise curve representations.

Note: Only the default *Pattern/Transparency* and *Line Style* are supported by Performance Graphics.

If you are using your mouse, you can make your palette selections (color, pattern and line style) simply by pointing at your choice with the cursor and clicking the left mouse button. You will see a square (probably black) surround the color, pattern or style that you just picked. This indicates that it is now the selected entry.

You can also use the keyboard to select from the palette. You should press the direction/arrow keys to move from color to color. As you press the direction keys, you will see a small square moving inside the color boxes. When the square is visible in the color, pattern or line style that you want, press Space. This has the same effect as pressing the left mouse button. The color that was indicated by the small box will be selected. Just like when you use the mouse, a larger square will appear surrounding your selection.

Using the keyboard to select from the palette works just as well as using the mouse. The only drawback is the extra time and keystrokes which are required to move the selection to the color you want.

Editing Current Colors

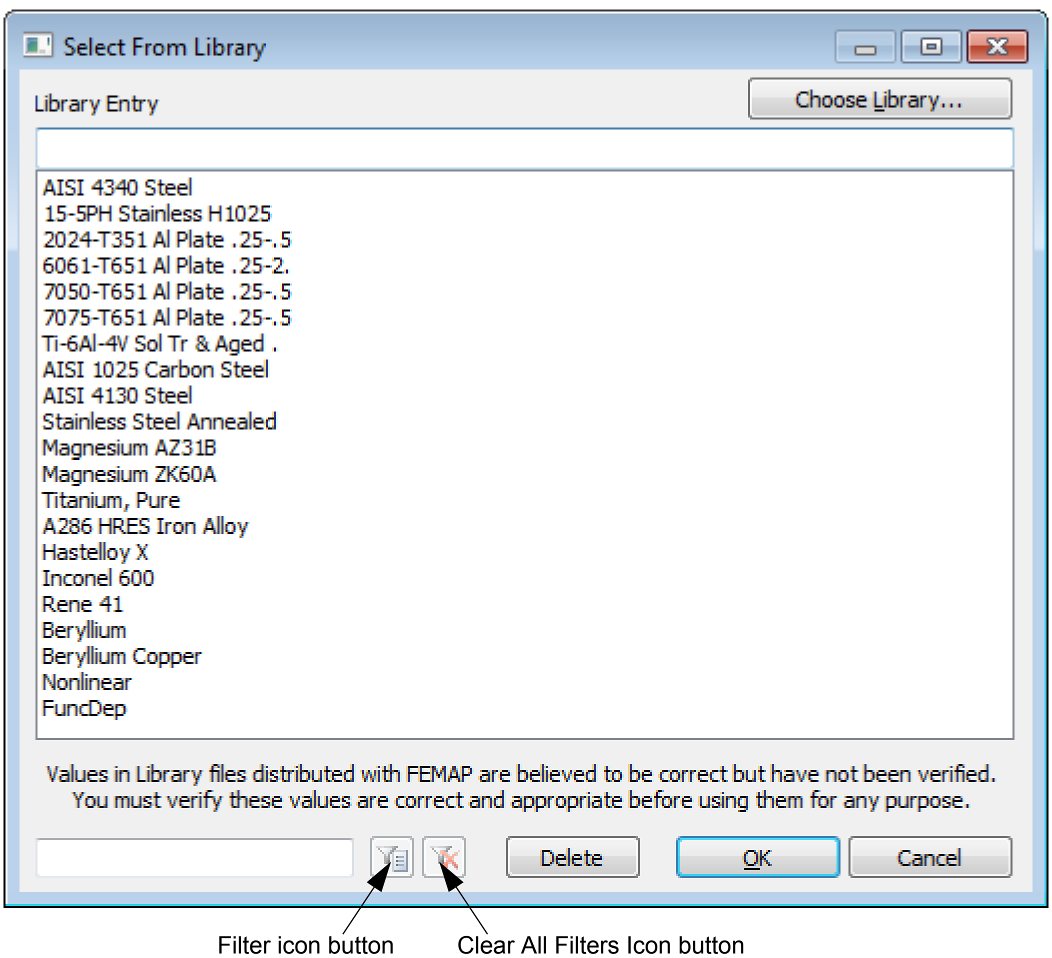
You can change FEMAP palette colors by selecting the color you want to change and then adjusting the red, green and blue values for that color. Choosing *Reset* will set the selected color back to its defaults. These color changes only apply to the current FEMAP model, and are not saved across models or even for the next time you work on the same model.

Working with Palette Libraries

Palette libraries overcome the limitations which were just described. By pressing the *Save* command button you can write the current palette, including any modifications you have made, to a file. In a future FEMAP session, or even a different model, you can press *Load* to restore your saved palette from the file. When you press either *Save* or *Load*, the standard file handling dialog box is displayed so you can choose a library file. The default file extension is *.PAL for all palette libraries, but you can specify any filename or filename extension. Unlike some other FEMAP libraries, only one palette can be stored per library file. You must therefore choose a new filename for each palette that you want to save.

4.3.6 Library Selection

When you are selecting materials, properties, views or other entities from a library you will see the *Select From Library* dialog box.



You can choose an entry from the library simply by choosing it from the list. In many cases however, the library may be large and you will want to search for a specific entry rather than looking through the entire list. In this case, enter any text that is found in the title you want and press the *Filter* icon button. The list will be reduced to just those entries that contain the text you specified. You can now enter additional text, and press *Filter* icon button again, to further reduce the list. Press *Clear All Filters* icon button to return to the full list and start again.

If you have multiple libraries for a particular entity type, you can quickly change from one library to another using the *Choose Library...* button and choosing a different *.esp file.

You can also remove a single entry from a library by simply highlighting it and clicking the *Delete* button.

The default library of each type is normally specified using the *File, Preferences* command and choosing the *Library/Startup* tab. You can always choose a different library there, but you can also pick a new library while you are working simply by pressing the *Library* button. You will then see the standard *File* selection dialog box where you can choose the library file that you want to use.

4.4 The Workplane and Other Tools

This section describes the workplane, as well as other tools for both graphical selection and numeric input.

4.4.1 The Workplane

Graphical selection on most models requires selecting entities from a 3-dimensional model from a 2-dimensional screen. FEMAP uses definition of a workplane to locate a 2-dimensional pick in 3-dimensional space.

When you make cursor selections or define two-dimensional geometry, the workplane is used to define the ultimate location in three-dimensional space. There are four methods of accessing the *Tools, Workplane* command to define the location and orientation of the workplane:

1. *Tools, Workplane* command
2. *Ctrl+W* shortcut key
3. Right mouse button - *Workplane*
4. Solid Toolbar

The last three shortcut methods allow you to redefine the workplane in the middle of another command. Using this technique, you can use the cursor to select one point projected onto one workplane, then realign the workplane for additional selections.

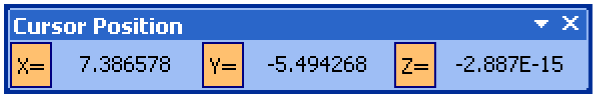
The workplane is a two dimensional plane which you can locate and align anywhere in three-dimensional space. By default, the origin of the workplane is at the global origin and the plane is aligned with the Global XY plane. When you make a graphical selection, the screen location which you selected is projected along a vector normal to the screen onto the workplane. The resulting three dimensional coordinates are located at the intersection of the projection vector and the workplane.

As stated above, the workplane can be aligned to any orientation. It is not restricted to be normal to the current view (although it can be easily set to that orientation). If you are using a workplane that is not normal to the current view, be careful when you make selections. As long as the workplane is not rotated too far from the screen normal you will have no problems accurately defining coordinates by picking. If the plane is rotated so that it is nearly “edge-on” to the view however, the projection of your screen location will be nearly parallel to the plane. The resulting intersection can have very large coordinates. In any case, picking with this alignment will be relatively imprecise.

If your workplane is exactly “edge-on” to the view, there would not be any intersection with the projection vector. In this case, FEMAP automatically projects onto a plane which is normal to the view, but which goes through the real workplane origin. This feature allows you to have multiple windows which are all displaying orthogonal views and still use all of them for selecting coordinates.

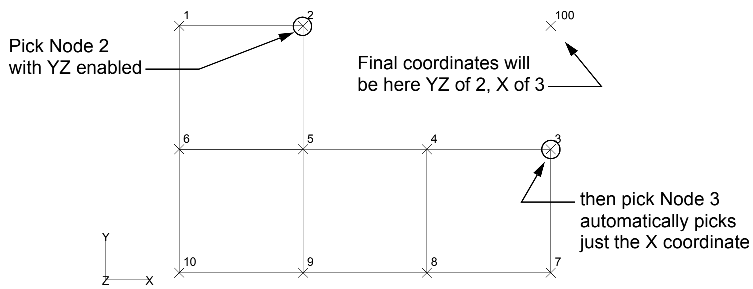
4.4.2 The Cursor Position Toolbar

FEMAP only fills in the dialog box coordinates after you make a selection. Until that time you can only tell the precise location of the cursor by its relationship to other entities drawn in your Graphics window. The *Tools, Toolbars, Cursor Position* command can provide more information on the cursor position, and provide additional graphical selection capability. Activating this command will display the *Cursor Position* toolbar.



With this dialog box active, as your cursor travels through any of the graphics windows, the displayed coordinates will be dynamically updated. You can make your graphical selections whenever you see the coordinates that you want.

In addition to displaying the coordinates, you will see three check boxes (X, Y and Z) which are all initially on. If you turn any of these boxes off (by clicking on them), you will notice that the corresponding coordinate disappears. In addition, when you make a graphical selection, only the coordinates which are enabled (on) will be selected and entered into the dialog box. Furthermore, after you make a selection with some coordinates disabled (off), the coordinates which were disabled are automatically enabled, and the coordinates which were enabled are automatically disabled.



This process is somewhat complex and is best explained with an example and a picture.

Suppose you want to select coordinates which match the Y and Z coordinates of Node 2 and the X coordinates of Node 3. With the cursor position displayed, turn off the X-coordinate and make sure that you are snapping to a node location. Then simply select Node 2 (this fills in the Y and Z coordinates of Node 2) and finally select Node 3 (this will fill in the X-coordinate of Node 3, since FEMAP automatically reversed the enabled/disabled coordinates after the first pick). If you need to do more complex selections involving all three coordinates you must enable/disable them manually, but it still only involves 1 or 2 clicks.

4.4.3 Snap To

The *Snap To* method of picking is a very powerful tool to locate your graphical selections at an exact position in the model. You may access this command, and/or change the *Snap To* method in five ways:

1. Right mouse button - *Smart Snap*, *Snap to Screen*, *Snap to Grid*, *Snap to Point*, and *Snap to Node*

2. Select Toolbar

3. Individual *Snap To* and *Smart Snap* shortcut keys

4. *Tools, Workplane, Snap Options* (*Smart Snap* not available)

5. *Ctrl+T* (when in another dialog box) (*Smart Snap* not available)

The last two methods allow you to change the snap mode, and to redefine the spacing and orientation of the snap grid. The first three methods allow you to simply change the snap mode. The shortcut keys (including *Ctrl+T*), the right mouse button, and the toolbar can be accessed while in other commands.

There are five modes available for the *Snap*, each with its own shortcut key for a quick change to the mode when you are inputting coordinates in a dialog box.

1. Smart Snap . . . . **Ctrl+R**

2. Snap To Screen (Snap Off) . . . . **Ctrl+S**

3. Snap To Node . . . . **Ctrl+N**

4. Snap To Point . . . . **Ctrl+P**

5. Snap To Grid . . . . **Ctrl+G**

If you simply want to change the snap mode, one of the preceding will enable you to change the snap mode immediately. They do not display any dialogs for further input. A message will be written to the *Messages* window, and the graphics cursor will change shape to identify the active snap mode.

By changing the Snap To mode, you can change the precision of your selection, specifically by snapping to previously defined Points or nodes with exact locations. This will enable you to obtain the preciseness you need for your operation, while still providing the ease and speed of graphical selection. This is extremely valuable when defining planes or vectors for such things as rotating and reflecting elements, where precise coordinate are required, and when nodes or points are already defined in appropriate locations. A brief description of each *Snap To* method is provided below.

Smart Snap

Picking snaps to the nearest node, point, midpoint of a curve, or center point of an arc based on proximity to cursor.

Note: *Smart Snap* always uses “Normal” pick mode to select coordinates, even when pick mode is set to “Query” or “Front”.

Snap to Screen (Snap Off):

This is the default mode. In this mode, no snapping is done. The location selected is based purely on the spot you pick in the Graphics window and, if you are picking coordinates, the position of the workplane.

Snap to Grid:

This mode uses an XY grid in the workplane. All cursor selections will be snapped to the closest grid point/line. Since you can control both the X and Y spacing of the grid points/lines, and the rotation of the grid in the workplane, you can use this method to round all cursor selections to the precision of the grid spacing. For example, if you specify a 1 inch spacing, all coordinate selections will be in increments of 1 inch from the origin of the workplane. Be careful if you are using this mode to select entities. Your pick is first snapped to the grid location, and then the closest entity is chosen.

For display purposes, you can change the grid to either dots or lines, or even make it invisible (not displayed). The style you choose has no effect on how the snapping is done.

Snap to Node:

This mode will adjust the location you select to the coordinates of the closest node. This mode is very useful if you need to reference your selections to other existing nodes. Be careful though if you are using this to create nodes. The one you create will be coincident. The same warning applies to picking IDs in this mode. Your selection will first be snapped to the node location, and then the closest entity will be chosen. You must have at least one node in your model, and it must currently be visible in the window where you make your selection to use this method.

Snap to Point:

This mode is identical to *Snap to Node*, except that the location is adjusted to the location of the closest point. You must have at least one point in your model, and it must currently be visible in the window where you make your selection to use this method.

When to Snap

By default, FEMAP will only use the snap mode that you choose when you are defining a coordinate. If you would like it to snap every time you pick in the Graphics window, use the *Tools, Workplane, Snap Options* command and turn off the *Coord Only* option.

4.4.4 Selecting Coordinates

Coordinates are defined throughout FEMAP for many purposes. In most cases you input coordinates through one of the standard coordinate, vector or plane dialog boxes described previously, but a few other dialog boxes do accept coordinate data. In any of these cases, you may supply the coordinates either by typing with the keyboard, or graphically selecting a location from any active graphics window.

To select coordinates graphically, follow the following steps:

1. Select any of the three (X, Y, Z) coordinate fields/controls.

2. Move the cursor to the desired location in the Graphics window

3. Press the left mouse button.

FEMAP will automatically fill in the coordinates which correspond to that location. Refer to the discussion of snap modes in the previous section for additional information regarding “snapping” the selected coordinates.

Normally when you select graphically, FEMAP will fill in the dialog box with the numerical coordinates of the location that you select. If you are snapping to a point or node however, FEMAP will insert equation functions. For example, if you snap to node 4, you would see XND(4), YND(4) and ZND(4). Similarly if you snap to a point, the XPT(), YPT(), and ZPT() functions are used. FEMAP uses these functions instead of the coordinate values to increase precision. When FEMAP loads the dialog box with a numerical value, those coordinates are only as precise as the number of digits in the dialog box. Typically, this is around six or seven significant digits. These functions reference the full, double-precision coordinates which are stored in the FEMAP database.

Since the six or seven significant digits is usually more accurate than you desire, you may want to disable this feature, so you can actually see the coordinate values. Just go to the *Tools, Snap To* command, and turn off the *Full Precision* feature. FEMAP will then always use the coordinate values, no matter how you snap. Turning *Full Precision* on will cause FEMAP to use the function references again.

4.4.5 Selecting Entities by their Titles

Many times, FEMAP will display a list of entities in a combo box. There are many ways that you can select entities from these lists:

- You can type an entity ID.
- You can select the entity graphically as described above.
- You can click the down-arrow (or the *Alt+Down* key) to view the list and select an entity.
- You could select the entity by typing its title.
- You can bring up a list of entities using *Ctrl + L*. This will only work if the entity you are selecting can have titles.

You have two choices to enter the title. You can either prefix (or enclose) the title with a single quote (') or a double quote (“). If you use a double quote, the title that you enter must exactly match the title of the entity. If you use a single quote, FEMAP will search all of the available titles and try to find the string that you enter. Any title that contains that string will be matched. Both methods are insensitive to case (i.e. Steel matches STEEL or steel). You can never select untitled entities using this method.

You will receive an error message if the title that you type does not match any of the entities in the list, or if it matches more than one. FEMAP will only make the selection if the title that you type uniquely identifies an entity. This restriction eliminates potential errors that could occur if FEMAP selected a different entity that happened to have a matching title.

4.4.6 Numerical Input - Real Number Formats

When you enter a real (floating point) number into a dialog box, FEMAP expects it to be in the International Number Format that you have chosen for Windows. Using the Windows *Control Panel*, if you pick the International option, you can set the Number (not Currency, Time or Date) Format that you want to use. FEMAP only uses the 1000 Separator (Thousands separator) and Decimal Separator settings. If you choose a 1000’s Separator, that character is simply ignored. You do not even have to enter it, but if you do, it will be skipped. The Decimal Separator, on the other hand, is used to defined the location of the decimal portion of the number. All numbers must be entered with the proper decimal separator, not necessarily “.”. The Leading Zero and Decimal Digits options are not used.

4.4.7 Numerical Input - The FEMAP Calculator

Any time you need to specify numeric input, whether it is a coordinate value or an ID, instead of simply typing the value, you can enter an equation. FEMAP will evaluate the equation and use the result as your value. Equations can consist of numeric values, variables, arithmetic operators (+,-,*,/), parentheses, and many other functions (SQRT(), SIN(), COS() and many more). For a description of available functions, see Section C, "Function Reference".

When you want to enter an equation, you can simply type the equation, or you can use the *Equation Editor*. You can access the *Equation Editor* by pushing *Ctrl+E* when you are in another command, and the *Equation Editor* will appear. It presents all existing variables, arithmetic operators and functions, and lets you preview the result prior to inserting the equation into the dialog control.

When using variables in an equation, you must predefine or create them using the *Tools, Variables* command before they can be used. When you want to use a variable in an equation simply precede the variable name by either an exclamation point (!) or an “at-sign” (@). The exclamation point (!) simply denotes the following character(s) as a variable, while the @variable_name operation allows an equation to reference the current value of another equation.

Using Variables

Variable Definition	Result
!variable_name	Uses the value of the variable when it was created or last updated.
@variable_name	Evaluates the equation which was used to define the variable, and uses the result of that equation.

Recursive Equations

The @variable_name operation allows an equation to reference the current value of another equation. When you use this capability, FEMAP must reevaluate all of the referenced equations. If you create multiple levels of equations, all using the @ operator, you can create a rudimentary "subroutine" capability, where you refer to the subroutine (an equation), simply by its variable name. We refer to this capability as a recursive equation because FEMAP must "recursively" reevaluate the resulting equation until it eliminates all of the @ operators. FEMAP allows you to create equations with up to 5 levels of "nested" @ operators. There is no limit on the number of @ operators in a single equation, just on the number of levels.

For example, you can define the following equations:

!x = 5*sin(45)
!a = 2.5*@x

and

!b = @x+@a

This is equivalent to typing (5*sin(45))+2.5*(5*sin(45)).

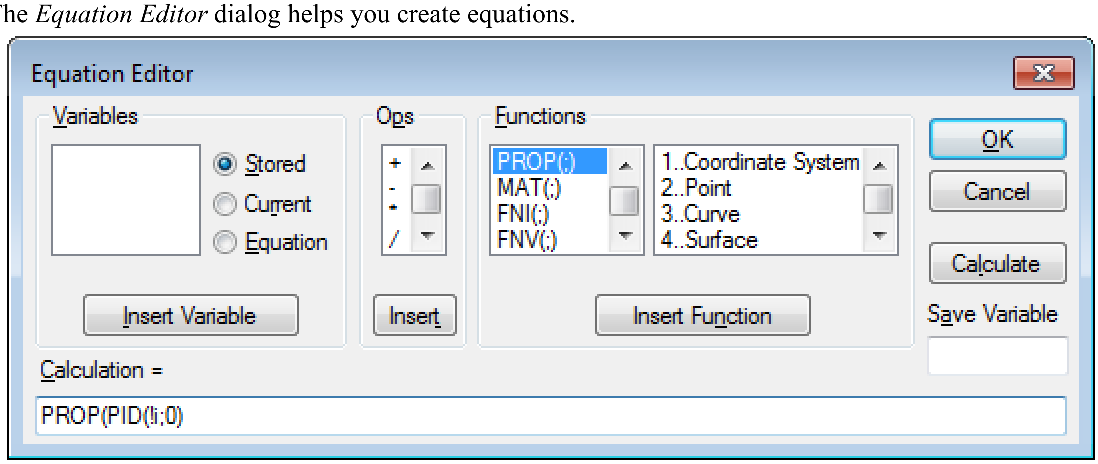
Note: Be careful not to create a situation where two variables reference each other using the @ operator. Evaluation of either variable would cause an infinite loop and will therefore fail when it reaches the limit of 5 nested operations. If you reach the nesting limit, either by this type of error or any other, FEMAP will display a series of error messages which represent a traceback of all of the evaluations that were taking place. You will have to repeatedly press *OK* to display these messages and continue.

Advanced Editing and Shortcut Keys

Windows provides extensive capabilities to enter and edit the text and numeric input which is required by FEMAP. You can use the various editing capabilities (direction keys, insert/delete,...) to create the input that you desire. You can also use other options such as copy (*Ctrl+Ins*) and paste (*Shift+Ins*) to duplicate the input from one dialog control into another control, or even to insert data from a different application. For controls that accept typed input (text boxes and drop-down lists) FEMAP supplements these basic Windows capabilities with the ability to display additional dialog boxes for advanced editing or entity selection, and the ability to execute certain commands. These additional capabilities are accessed through keyboard shortcut keys or the *Quick Access* menu described above.

4.4.8 Equation Editor - Ctrl+E

Whenever you need to enter numeric input, you can always enter an equation in place of the actual numeric result. The *Equation Editor* dialog helps you create equations.



First, it contains a much wider edit control so you can simultaneously see much more of the equation text. More importantly however, it presents all existing variables, arithmetic operators and functions, and lets you preview the result prior to inserting the equation into the dialog control.

Variables:

This shows a list of all of the variables which are defined in the current model. When a variable is created, both the defining equation and the result of that equation are stored. Choosing *Insert Variable* will modify the current equation using the selected variable and variable option. If “Stored” is selected, !variable_name will be inserted. When the equation is evaluated, this will use the stored numeric value of the variable. If *Current* is selected, @variable_name will be inserted. In this case, the stored defining equation will be reevaluated when the new equation is evaluated, and the new value will be used. If *Equation* is selected, the entire defining equation will be inserted. This will let you view and modify the equation. If the variable to be inserted was defined using a simple numeric value, then all of these options will have the same result.

Ops:

This section simply allows you to see and insert a list of the available arithmetic operators. Using this option, parentheses are always inserted in pairs and balanced.

Functions:

In addition to numbers, FEMAP equations can contain arithmetic, trigonometric and model query functions, all of which can be inserted using this list. Some of the model query functions require an argument which is an entity type number. They are all shown with a “->” in the function list. For those functions, the argument is automatically inserted based on the entity type selected from the second list. For more detailed descriptions of each function, see Section C, "Function Reference".

Note: Please be very careful when using the SQR and SQRT functions in different portions of FEMAP. When working within the FEMAP interface, such as creating an equation for loading, SQR is “square”, while SQRT returns the “square root”. When creating a “script” using the *API Programming* window (see Section 7.2.10, "Tools, Programming, API Programming"), SQR will actually return the “square root”, not “square” the value.

Calculate:

This button will automatically evaluate the equation that you are defining and display the result.

Save Variable:

If you want to save this equation (and its result) as a variable, simply enter the name of a new variable in this text control. Then if you choose *OK*, the variable will be created.

If you are using your mouse with the equation editor, you do not have to press the various Insert buttons. Instead, you can simply double-click with the left mouse button in any of the lists. The entry that you are pointing at will be inserted into the equation.

The FEA Process

5

This topic gives a general overview of the steps used to create a finite element analysis model. There are descriptions of some commands and processes for creating geometry, elements, materials and properties, loads and constraints, and other viewing and model manipulation commands used in FEMAP. This topic is just an overview of the process. For in-depth information on all FEMAP commands, see *FEMAP Commands*. You may also refer to the *FEMAP Examples* for sample step by step instruction in building, using and manipulating models.

5.1 Geometry

Geometry for FEA is different than most other modeling applications. The only reason for creating geometry in FEMAP is so you can more easily generate an accurate mesh. Keep this in mind when creating models that may be used for FEA. An example of how FEA geometry may differ from the actual part can be as simple as a corner on a part.

Good engineering practice dictates that a corner be filleted, to relieve stress concentrations and to match the radius of the cutting tool being used to manufacture the part. However, a small fillet in FEA can significantly complicate the problem. Accuracy in FEA depends on element size and aspect ratio, and an efficient transition between elements of different sizes. It takes a very small mesh and many more elements in the area of the fillet to properly analyze it. It is much easier and much faster to leave the corner as a corner and use the stress concentration that appears there as an upper bound. If an area is so critical that the fillet or whatever other feature you are modeling must remain, take care to create a good mesh.

5.1.1 Methods and Snap To

As discussed in Section 4, "User Interface", many FEMAP dialog boxes contain a *Method* button. This button allows you to access a drop-down menu that can be used to change the way you specify coordinate locations, as well as other information. It allows you to choose the way you want to define a location, vector, or plane. There are many more options depending on which command you are using and what geometry you have created. Always check the Method menu first when you think there should be easier ways to define locations, vectors or planes because most likely there are.

*C*hanging your *Snap To* method can save you significant time and effort, especially in creating geometry. *Snap To* sets the cursor mode. It can be set to snap to the screen (snap off), to nodes, to points or to the grid. It can be changed at any time with the shortcut (right mouse) menu or the View toolbar icons. It is especially valuable to change the *Snap To* mode when defining vectors or planes. You could actually define a plane by defining three locations by snapping to a point in space (screen), a node, and a point.

Note: Remember to change your *Snap To* mode when you have nodes and points already defined that are in appropriate positions to define vectors and planes. It will save you considerable time by replacing the keyboard coordinate input.

5.1.2 The Workplane (2-D and 3-D Geometry)

The workplane allows you to create two dimensional geometry in a three dimensional world. The workplane is a user-defined plane in FEMAP on which the results of certain commands will be placed. The workplane becomes very important when generating geometry.

When creating geometry, you have the option to work in 2-D or 3-D space. The geometry creation commands for the basic entities such as line, arc, circle, and spline are separated into two major sections, 2-D and 3-D. Each submenu for these creation commands are divided into a top section, which is 2-D, and a bottom section which is 3-D.

Note: When you use a command that is above the separator line in these Geometry commands, the entity will always be created in the workplane. Any coordinates you define, if not already in the workplane, will be projected onto the workplane.

If your line is not drawn where you expected it to be, most likely it has been projected onto the workplane. All commands below the separator line perform operations in 3-D so your coordinate inputs will be used without modification.

Geometry creation for a large 3-D model can seem like a difficult task, especially when you are new at modeling. However, most parts can be created by visualizing them as a series of 2-D sections. Furthermore, many individuals have difficulty picturing objects in 3-D when viewing inherently 2-D monitors. For this reason, it is important that you become familiar with moving your workplane so you may work in a series of 2-D steps and simplify the model creation process.

You may also want to align the workplane to your current view to coordinate the viewing/creation process. This is simply done by using *Tools, Workplane*. Pick *Select Plane*, click on the *Method* button, and change the *Method* to *Align to View*, and provide the appropriate data. You may also align the view to the workplane with *View, Align By, Workplane.*

You may also define a new workplane based upon its relative position to the current workplane. The *Move Plane* section of the *Tools, Workplane* command enables you to define the new workplane by an offset translation and/or a rotation from the current workplane. You may move the workplane in its Z direction and rotate it around its Z axis. This is a quick way to change the location and azimuth of the workplane without having to define three new points.

Another method for defining a workplane is using an existing surface. You first pick a surface, the normal of which is used to define the normal of the plane. Then define a point to use as the origin. The normal of the surface and the origin point completely define the plane. You may also define the X and Y directions on the workplane. You pick a point that will be projected onto the plane to define the X direction and the Y direction will be the cross product of this X direction and the normal from the origin.

5.1.3 Basics - Points, Lines and Curves

FEMAP gives you many options for creating points, lines and curves. These options are contained under the *Geometry* command. Points, lines, and curves are generally the starting blocks for any model, and therefore it is important to have a good understanding of the different creation methods. The simplest method to create a point is to define its coordinates. However, by pressing the *Method* button, you can access 15 different methods of defining that coordinate location. Some of these methods will not be available if the required entities do not exist (i.e. you cannot use the *Onto Surface* method if you have no surfaces in your model). Simply select the most appropriate *Method f*or your circumstance and input your values.

There are also a large number of ways to create a line. The four most basic are horizontal, vertical, points and coordinates. Points and coordinates differ in that *Point* commands create a line between existing points and the *Coordinates* command will create a line between any two specified locations. FEMAP will automatically generate points at the end of all lines during the creation process. When creating a line with either the *Geometry, Curve-Line, Points* or the *Geometry, Curve-Line Coordinates,* command, you will create a line in 3-D space. When using a command such as *Geometry, Curve-Line, Coordinates,* remember that you may still use the *Method* button to access other ways to input the coordinates, exactly as you would if you were creating a point.

Horizontal and vertical lines are created to a length specified under the parameters of the line at a location on the workplane (horizontal is along the x-axis of the workplane and vertical is along the y-axis of the workplane). Remember, commands above the line are created on the workplane, and those below it are created in 3-D space. Other commands under *Geometry, Curve-Line* enable you to create lines by inputting their relationship to other curves or points in your model.

Arcs and circles can also be created in the workplane or in 3-D space using a variety of commands. All arcs are typically created by specifying three entities such as center - start - end, start - end - radius, three points etc. Arcs in the workplane are drawn as positive in the counter clockwise direction (input of a negative angle when using an angle as one of the inputs will cause FEMAP to draw a clockwise arc). 3-D arcs have no convention. Their direction will be specified in other ways. All of the methods can be used to create equivalent arcs. The various commands simply ease the input process. Once again, when specifying coordinates, you can change the method of specification to further simplify the input.

Circles are created in much the same way as arcs except, of course, they are complete circles. Again, they can be created in the workplane or in 3-D space. Most methods are self explanatory. For more details, see Section 3.2, "Creating Curves".

5.1.4 Splines

Splines are complex curves of at least four points. In FEMAP splines of four points as well as those created with the ellipse, parabola, hyperbola, equation, tangents and blend options will be stored as cubic Bezier curves. All other splines will be stored as B-splines. The actual curve of the spline will pass through the first and last control points but not through the others. The other points influence the curvature of the spline. The farther a control point is from the previous control point the more the spline is 'pulled' in that direction. Splines can also be created on the workplane or in 3-D space. A number of methods are available, the simplest being *Geometry, Curve-Spline, Points,* where you select 4 to 110 points on the spline and the control points are automatically calculated.

Note: The *Cancel* button on the dialog box is utilized to both cancel the creation of the spline as well as create it. If less than four points have been chosen, the *Cancel* button will enable you to terminate the process without creating a spline. Once four points have been defined, however the *Cancel* button is used to terminate input of more points and a spline is created. If you make an input error after four points have been defined, you cannot cancel the procedure without creating a spline. Simply use the *Tools, Undo* command to remove the spline if it is inaccurate. This is true for all procedures under *Geometry, Curve-Spline* that enable you to create B-splines.

5.1.5 Curves from Surfaces

The curves from surfaces are only available when the Parasolid solid modeling engine is active. There are 11 different methods to obtain curves from existing surface data and two modes available. If update surfaces is checked, new curves will break the surfaces with which they interact, essentially imprinting onto the surfaces. If update surfaces is Off, they are simply curves.

The 11 types of curves from surfaces are listed below:

- *Intersect* - Create curves at the intersection of two surfaces.
- *Project* - Project a curve onto a surface using the point on the surface closest to the point on the curve.
- *Project Along Vector* - project a curve onto a surface using a specified vector.
- *Parametric Curve* - Create a curve from a constant U or V parametric value of a surface.
- *Slice* - Create a curve at the intersection of the surface and a specified cutting plane.
- *Split at locations* - Create a curve from one location to a second location following the underlying surface.
- *Offset Curves/Washer* - Create curves around holes or selected curves (edges, slots, etc.).
- *Pad* - Create a square pattern around a hole which is broken into four equal regions.
- *Point to Point* - Create a curve from one existing point to another following the surface.
- *Point to Edge* - Create a curve from an existing point to the closest point on a selected curve.
- *Edge to Edge* - Create curves from the end points of selected curve(s) to closest locations on a different selected curve.

5.1.6 Modifying the Basics

If you are creating geometry directly in FEMAP, use the *Geometry* menu commands to generate the original geometry, but the *Modify* commands allow you to both change original geometry, as well as create new geometry between existing entities (such as fillets).

The *Modify* menu has three separate sections. The following topic focuses on the top section, the commands for modifying lines and curves in this section. The following six commands are available to quickly modify geometry:

- *Trim* - cuts curves at the locations where they intersect other curves.
- *Extend* - moves the midpoints of one or more curves to a specified location.
- *Break* - splits one or more curves into two pieces at a location you specify.
- *Join* - extends or shortens two selected curves to their intersection.
- *Fillet* - connects two curves with an arc of a specified radius.
- *Chamfer* - trims two intersecting curves at a specified distance from their endpoints, and connects the trimmed area with a new line.

These commands can be used to quickly change a model from a set of intersecting and overlapping lines to an accurate representation of your part. In fact, once you are familiar with these commands, you can start your model with lines in the proper directions, and simply trim, fillet, etc. until your model is complete. For example, it can be much faster to draw the outline of the part with straight lines and then fillet where required, rather than producing each individual arc with the *Geometry, Curve-Arc* command.

The second section of the *Modify* menu command allows you to move objects, including geometry. You can *Project, Move To* a point, *Move By* a vector, *Rotate To* a point, *Rotate By* an angle, *Align* or *Scale.* These commands can operate on coordinate systems, points, curves, surfaces, volumes, solids, nodes, and elements. Moving one entity will automatically move all associated entities. For example, moving a curve will also move all points connected to that curve but not those coincident to it. You may also move an entire mesh by moving the coordinate system that define the nodes.

You may make copies of existing entities utilizing commands under the *Geometry* menu. You can make copies of points, curves, surfaces, volumes, and solids. You can copy along a vector, in a radial direction, by rotating around a vector, reflecting across a plane, and scaling from a location. The procedures for executing the above commands are straightforward and the exercises in the *Examples* guide will show you the usefulness of many of these commands.

Note: The *Modify Trim, Extend,* etc. commands are not available for solid geometry curves. These curves must be manipulated with the *Geometry, Solid* and *Curve from Surface* commands.

5.1.7 Surfaces, Boundary Surfaces, Volumes, Solids

For all models the ultimate goal of the preprocessing portion of FEA is meshing. For most models you will use surfaces, volumes, solids and boundary surfaces to create the mesh. Therefore, it is important to have a good understanding of how they work. Most often you will create these entities from already existing geometry (surfaces and boundary surfaces from lines and curves, volumes from surfaces, solids from surfaces and boundary surfaces). Surfaces (including boundary surfaces) are used to create 2-D elements and volumes and solids are used for 3-D elements.

5.1.8 Surfaces

There are several general methods to create a surface:

- Select 3 or 4 corners and a planar surface (tri or quad) will be created between them.
- Use existing curves to create a surface from “bounding” curves.

  Edge Curves - three or four bounding curves coincidentally ended.

  Aligned - four control curves aligned in the same parametric direction.

  Ruled - create a ruled surface between two curves.
- Move a curve along a path.

  Extrude - straight line path

  Revolve - uses an angle around a center vector.

  Sweep - follows the path of a chosen curve.
- Analytical (Predefined Shape) - planar, cylindrical, conical, tubular or spherical surfaces.
- Offset - creates a copy of an existing surface and locates it using specified offset from original surface.
- Convert - attempts to convert a boundary surface into a parasolid surface.
- Remove Hole - removes internal loop (holes, slots, cutouts) from a surface.
- Create “General Bodies” of NonManifold Geometry

  NonManifold Add - adds nonmanifold geometry (typically surfaces at “t-junctions” relative to one another) together into a one “general body”.

  Recover Manifold Geometry - Creates several manifold geometric bodies from a single “general body”.
- From Mesh - creates a surface by selecting shell elements from an existing mesh.
- Using existing solid geometry.

Explode - create individual surfaces for all faces of a solid. The solid is lost.

Midsurfaces - create midsurfaces between surfaces of thin-walled solids.

Four sided surfaces are considered optimum for meshing purposes because you can easily generate a nicely mapped mesh of planar elements. During the meshing of surfaces or solids (which mesh the surfaces first) FEMAP will determine which surfaces can be map meshed and will do so accordingly. You can also use the *Mesh, Mesh Control, Approach On Surface* command to dictate a mapped mesh on a surface.

5.1.9 Boundary Surfaces

You may use the *Geometry, Boundary Surface* command to create a boundary. A series of lines and curves with coincident endpoints are selected. Holes can be added by picking existing curves inside the boundary curves that form closed holes.

You may also create a boundary automatically by using the *Sketch* command. The *Sketch* command will allow you to use the *Geometry* creation commands to draw lines, arcs, etc. When you press *Finish Sketch,* FEMAP will automatically take these curves and produce a boundary. This is a very convenient method to quickly define a boundary.

Boundaries are created from any number of continuous curves. These curves must be either joined at the ends or have coincident points and be fully enclosed. They cannot just intersect.

Boundaries can contain holes, as long as the area of the hole is completely contained within the boundary and they do not overlap. FEMAP will automatically determine which curves if any represent holes in the boundary. Because of the arbitrary geometric nature of boundaries, many models may require you to be more careful in the mesh generation process to obtain a good mesh. For more information on the boundary mesher, see Section 5.2, "Elements and Meshing".

5.1.9.1 Volumes

There are three basic methods to create volumes. They are:

- Multiple existing entities as components

  Corners - locate four to eight corners.

  Surface - four to six bounding coincident edged surfaces.

  Between - two surfaces of the same shape.

- Move a surface along a path:

  Extrude - straight line.

  Revolve - angle around a central vector.

- Analytical (predefined shapes) - cylinder, cone, tube, or sphere.

The only reason to create a volume is to create a mesh. If you can create a volume that accurately represents your part, you can readily create a good mesh. However, volumes have two important restrictions:

- No more than six surfaces or eight corner points can be used to define a volume.
- A volume must be continuous. No voids are allowed.

These restrictions limit the usefulness of volume meshing. For this reason, this manual will concentrate on other methods to obtaining 3-D meshing including the *Mesh, Extrude*, the *Mesh, Revolve*, and the solid meshing commands. They all have the characteristic capability to create a solid element mesh from a 2-D mesh. Since it is impossible to obtain a good 3-D mesh by starting with a bad 2-D mesh, it is even more important that you understand how to generate good 2-D meshes. The mesh generation topic will go more fully into this aspect of FEA.

5.1.10 Solids

The solid meshing commands are available in all configurations of FEMAP. They allow you to create solid models in the Parasolid Solid Engine. You may also import solid models created in other CAD programs using the Parasolid engine, and then modify or mesh them using FEMAP. There are additional options that allows you to import IGES trimmed surface data that can be stitched into a FEMAP solid, or import STEP AP203 solid data.

In FEMAP there are two basic ways to create solids:

- Using primitives - Create blocks, cylinders, cones, and spheres.

- Using surfaces/boundaries - *Extrude/Revolve* to create a new solid or *Add/Remove* material from an existing solid. *Sweep* to create a solid which follows a "drive curve" or *Sweep Between* to create a solid between two selected surfaces. *Stitch* to create a solid from surfaces that completely enclose a volume.

There are also a number of ways to modify existing solids.

- *Fillet* - Fillet an edge/edges of a solid with a specified radius.
- *Chamfer* - Chamfer an edge/edges of an existing solid to a specified length.
- *Shell* - Convert a solid to a thin walled shell by offsetting faces.
- *Thicken* - Thicken Sheet bodies into solids or increase/decrease thickness of solids.
- *Extend* - Choose a face on one solid and extend it to the chosen face of a different solid.
- *Fill Hole* - Choose a face of a cavity and a new solid will be created to "fill" the entire cavity.
- *Remove Face* - Choose faces to remove from geometry to de-feature the solid.
- *Add* - Join two solids to form a single solid.
- *Remove* - Subtract one solid from another.
- *Common* - Create a solid from the intersecting volumes of two solids.
- *Embed* - Create two solids, one from solid from the intersecting volumes of two solids
- *Intersect* - Intersects the surfaces of the selected solids.
- *Slice* - Cut a solid with a specified plane; with a specified plane, but leave matching surfaces on both solids: with existing face(s) of sheet solids; or with existing curves using the normal vector or a specified vector.
- *Embed Face* - Extrude a face into a new solid and embed it into the existing one.

Three utility commands exist for solid modeling.

- *Stitch* - Sew surfaces into a FEMAP solid. Particularly useful for IGES files.
- *Explode* - Explode a solid into individual surfaces created from each face. The original solid data will be lost.
- *Cleanup* - Remove extra curves/points that are not required to define the solid.

5.2 Elements and Meshing

As mentioned above, the entire reason for creating geometry is to produce a good finite element mesh. This section describes the different element types contained in FEMAP, as well as meshing procedures to obtain these elements.

5.2.1 Element Types

There are four basic element groups in FEMAP. They are line, plane, volume and "others". A list of all the elements currently supported by FEMAP, including a brief description, is provided below.

5.2.1.1 Line Elements

- **Rod** - Uniaxial element with tension, compression and torsional stiffness. No bending or shear. Typically used to model trusses.
- **Tube** - Rod element with tubular cross section. Some analysis programs will support bending and shear. Often used to model pipes.
- **Curved Tube** - Tube element with an arc for the neutral axis.
- **Bar** - Uniaxial element with tension, compression, torsion and bending. Used to model general beam/frame structures. Similar to beam.
- **Beam** - Uniaxial element with tension, compression, torsion and bending. It can be tapered and have different properties at each end. Used to model beam/frame structures.
- **Link** - Rigid link with six stiffnesses at each end. Used to represent members that are very stiff compared to their connections.
- **Curved Beam** - Beam element with an arc for the neutral axis.

- **Spring/Damper** - Stiffness and damper element. Can be torsional or axial. Used to represent purely torsional or axial structural members. Also, used to create CBUSH element in Nastran.
- **DOF Spring** - Spring element used to connect any one degree of freedom from one node to any one degree of freedom of another node with a specified stiffness.
- **Gap** - Nonlinear element with different tension, compression and shear stiffness. Used to represent surfaces or points which can separate, close or slide relative to each other.
- **Plot Only** - Nonstructural. Used to represent structural features that are not being analyzed but aid in the visualization of the model. Also used to define ABAQUS rigid elements for contact.

5.2.1.2 Plane Elements

- **Shear Panel** - Resists only shear forces. Used to model structures which contain very thin elastic sheets, typically supported by stiffeners.
- **Membrane** - Resists only in plane normal forces. Used to represent very thin elastic sheets.
- **Bending Only** - Resists only bending forces. Used to model plates that will only resist bending.
- **Plate** - Resists membrane, shear and bending forces. Used to model structures comprised of thin plate shells.
- **Laminate** - similar to the plate element, except that this element is composed of one or more layers (lamina). Each layer can represent a different material. To create a laminate you need a *Layup* to specify the material, thickness, orientation angle and global ply ID (optional) of each ply and a *Laminate* property.
- **Plane Strain** - Biaxial plane element. Create a 2-D model of a solid which does not vary through its depth. Used to model very thick solids which have a constant cross section.
- **Axisymmetric Shell** - 1 dimensional element used to represent surfaces of revolution.
- **Planar Plot Only** - Nonstructural. Used to represent structural features that are not being analyzed but aid in the visualization of the model. Also used to define ABAQUS rigid elements for contact.

5.2.1.3 Volume Elements

- **Axisymmetric** - Two dimensional element used to represent volumes of revolution.
- **Solid** - Three dimensional solid element used to represent any three dimensional structure.
- **Solid Laminate** - Similar to three-dimensional solid element, except that this element is composed of one or more layers (lamina). Each layer can represent a different material. To create a solid laminate you need a *Layup* to specify the material, thickness, orientation angle and global ply ID of each ply and a *Solid Laminate* property.

5.2.1.4 Other Elements

- **Mass** - Three dimensional mass and/or inertia element located at a node. Used to represent parts of a structure which contain mass but do not add stiffness.
- **Mass Matrix** - Generalized mass element. Mass and inertia properties are defined as a 6x6 mass matrix.
- **Spring/Damper to Ground** - Used to create CBUSH element on a single node connected to “ground”.
- **DOF Spring to Ground** - Spring element to connects any one degree of freedom to “ground”.
- **Rigid** - Rigid connection between a master and unlimited number of slave nodes. Used to model connections which are very stiff compared to the rest of the model.
- **General Matrix** - General stiffness, damping, or mass element defined by a 6x6 or 12x12 stiffness matrix. Models custom stiffness connections not adequately represented by other stiffness elements.
- **Slide Line** - Contact element which allows input of frictional and stiffness contact information between nodes and surfaces. Modeling of finite sliding surface interaction between two deformable bodies.
- **Weld/Fastener** - Connection element between two sets of shell elements which uses weld diameter, length, and an isotropic material to determine the stiffness of the connection. Can also be used to simulate “Spot” welds. Fastener elements are available for certain solvers which allow several options for modeling specific behavior.

5.2.1.5 Analysis Output

Check the element reference and the translator reference to determine which elements are supported and how they are translated for your analysis program before you create them.

5.2.1.6 Mesh Sizing

Before you create elements, you should first determine the mesh size using the *Mesh, Mesh Control* command. You can set a default mesh size or default number of elements, which is used for all geometry where a specific size or number of elements is not defined. You can also define a specific mesh size or number of elements along a line or on a surface.

Mesh sizes can also be biased so that a finer mesh can be obtained at either end or in the middle. Mesh sizes can be set interactively. You can also define hard points on curves or surfaces to ensure a node is placed at that location. You can even define a particular mapped meshing approach on a surface.

Always define mesh sizes carefully to ensure good element aspect ratios, high resolution in areas of large stress gradients and proper matching of nodes where curves, surfaces, boundaries or volumes/solids meet. The last point is especially important because if nodes are not coincident, your model will have free edges or faces at these points and will not solid mesh or solve properly. Remember to always merge coincident nodes before attempting to solid mesh or analyze your model.

5.2.1.7 Mesh Attributes

If you are meshing geometry with different element types or properties you may find it helpful to set meshing attributes. These commands allow you to specify various meshing parameters directly on geometry in FEMAP. This can save you time by allowing you to select multiple entities to mesh at the same time while still meshing with different parameters. Parameters that can be set include: properties, offsets, releases, orientations.

5.2.2 Element Creation

In FEMAP there are seven methods to create elements:

1. Create an element one at a time - *Model, Element* command.

Useful for simple geometry, line elements, and to fix areas of distorted elements.

2. Create multiple elements on geometry (curves, surfaces, solids and volumes).

Mesh, Geometry - line elements on curves, plane (or axisymmetric) elements on surfaces, and solid elements in volumes and solids. When meshing surfaces, you can also combine multiple surfaces by creating a multi-surface boundary which will be meshed to ignore interior features.

3. Copy existing elements.

Mesh, Copy - make copies of existing elements along a vector.

Mesh, Radial Copy - similar to copy except in a radial direction.

Mesh, Scale - create a scaled copy of the element around a given location.

Mesh, Rotate - rotate the duplicate copies around a vector (axis of rotation).

Mesh, Reflect - reflect (or flip) elements through a plane.

4. Convert 2-D model (curves or elements) into a 3-D model of planar or solid elements.

Mesh, Extrude - often used to generate 3-D solids from a 2-D mesh.

Curve - creates 2-D elements by moving existing curves along a vector or curve.

Element - creates 2-D from 1-D and 3-D from 2-D elements by moving along a vector, a curve, or element normals.

Mesh, Revolve - similar to *Mesh, Extrude,* except revolves around a vector. Often used to solid mesh volumes of revolution.

5. Use non-geometry meshing commands.

Mesh, Between - produces meshes (1-D, 2-D or 3-D) between corners.

Mesh, Region - creates a ruled region of nodes and/or elements between patterns of nodes.

Mesh, Transition - produces an automatic mesh between existing nodes. Useful to “fix” regions between surface meshes that are improperly connected.

Mesh, Connect - zip or unzip elements at the nodes. Connect with line elements, rigid elements, or constraint equations.

Mesh, Editing - split existing elements with pre-defined patterns, interactively, based on multiple selection, or nodal connectivity to specific element.

Mesh, Remesh/Smooth - used to modify an existing mesh. Useful for fixing or "cleaning-up" a distorted area of a mesh.

Mesh, Edge Members - creates 1-D elements from 2-D or 2-D elements from 3-D using selected nodes found on the edges of selected elements.

6. Use solid meshing - automatic meshing with 3-D tetrahedral elements.

Most useful when a 3-D solid mesh of a fairly complex geometry is required.

Mesh a solid created in FEMAP with the Parasolid modeling engine.

Automatically mesh any enclosed volume of planar elements.

Import and mesh geometry from any ACIS or Parasolid-based CAD package.

Import, stitch and mesh IGES trimmed surface data.

Import STEP solid body entities and mesh automatically.

The solid mesher also incorporates the capability to import a triangular surface mesh from a Stereolithography file. The triangular surface representation found in most STL files is not of sufficient quality (element shape and aspect ratio) to be fed directly into the automatic mesher. The *Mesh, Remesh* menu contains commands which can help you transform the poor triangular surface mesh into a better one.

7. Use solid meshing - - semi-automatic meshing with 3-D hexahedral elements.

Useful for creating partial or full solid hexahedral meshes.

Subdividing of solid into hex meshable regions is required.

Mesh sizes on all solids to be hex meshed must be set at the same time using the *Mesh, Mesh Control, Size On Solid* command with the hex meshing option chosen. Matching surfaces are linked and mesh sizes set so the hex mesh can propagate between solids.

5.2.2.1 Surface Meshing Guidelines

The mesh generation tools above provide a wide array of methods to generate your mesh. Examine your part before you begin the meshing process to determine which method is most applicable to your part. The guidelines below provide a few handy tips for the mesh generation process for surface element meshing.

- Most meshes involve creating geometry first. Define these accurately from the beginning, keeping in mind you are using it for meshing purposes only (i.e., remove small features if they are not critical to the analysis).
- Use the *Geometry, Surface* command to create four(4)-sided surfaces whenever possible, specifically for critical stress areas. Subdivide your part into regions if required. Four-sided surfaces enable an all quad mapped mesh with little or no distortion.
- Use the *Geometry, Boundary Surface* command to define boundaries that cannot be generated as surfaces. Remember the boundary mesher will work best with areas that have similar length and width dimensions (globular as opposed to long and thin).
- If you have solid geometry that has surfaces that are highly skewed, or you just have surfaces that are split at places that you do not want to split the mesh, use the *Geometry, Boundary Surface, From Surfaces on Solid* command to create a multi-surface boundary. This boundary surface will then be meshed, and will ignore the "interior" curves and other features. Many surface models will generate much better meshes using this approach.
- Define your default mesh size before you start meshing by using *Mesh, Mesh Control, Default Size.*
- Use the *Mesh, Mesh Control, Size on Curve/Surface* command to individually define mesh sizes for curves and surfaces that are used in more than one mesh region. Do this before you start meshing to prevent misalignment between meshes in your model.
- Once mesh sizes are established, use *Mesh, Geometry, Surface/Solid* commands to mesh your model. When performing a free/boundary mesh, take note of allowable distortion for quad elements. You may want to change the default to allow more or less distortion.

- Use *Mesh Control, Approach on Surface* to link surfaces or specify mapped meshes on surfaces that would otherwise be free meshed.
- You can use the *Mesh, Revolve/Extrude* command to generate 2-D elements from 1-D elements or curves whenever possible. This can be useful for cylindrical shapes.
- Utilize symmetry whenever possible to reduce meshing effort. Model size is significantly reduced (and therefore run time) if the loading/constraints are also symmetrical. If loads/constraints are not symmetrical, you can use the *Mesh, Reflect Element* command to reflect the mesh through a plane.
- Remember, you may also want to use *Mesh, Copy* and *Mesh, Rotate* to produce replica elements instead of performing more surface or boundary meshing.
- Use the *Tools, Check, Coincident Nodes* command to merge coincident nodes and connect the meshes.
- Use the *View Select* command and change the *View* to *Free Edge* to verify that you do not have any unwanted free edges in the model.

The above guidelines provide a good basis for surface element meshing. It is critical, even when solid meshing is the ultimate goal, that you establish a good surface mesh.

5.2.2.2 Solid Meshing Guidelines

- Often times you can avoid using volumes or solids by simply extruding or revolving truly planar elements into solid elements. If your part has a consistent third dimension, use *Mesh, Extrude* or *Mesh, Revolve* to create solid elements.
- Use the *Mesh, Edge Members* command if planar elements are required on faces of solid elements. Once planar elements are created, you can extrude them into solid elements.
- For simple solid parts, use volume meshing, *Mesh, Between*, or *Mesh, Region* to create a solid mesh. These procedures cannot be used, however, if there are voids in the volume.
- If you have solid models with holes or other complicated intricacies, use the solid tetrahedral mesher. This mesher creates a surface mesh first, so all items applicable to surface meshing apply. If you have purchased FEMAP you may import in ACIS, Parasolid, IGES, STEP, or STL files or use the FEMAP solid modeling commands to define geometry to create 3-D meshes.
- If you do frequent hexahedral meshing, become familiar with the types of solid shapes that can be hexahedral meshed, and focus on slicing your solid models into shapes of those types. When slicing your solid, take care to avoid creating sliver surfaces or solids.
- There are a wide array of solid and surface modification and combining tools. Take the time to learn what each one does. Used in combination they can be very powerful and accomplish many different tasks useful for solid meshing preparation. In addition, as a first step before solid tetrahedral meshing, try using *Mesh, Geometry Preparation* to “prepare” geometry using a combination of “smart” surface splitting, feature suppression, and creation of combined curves/boundary surfaces.
- Use the *Explode* command to create surfaces that you can cleanup, and then stitch back into a solid for meshing.

If you follow the above guidelines for both surface and solid meshing, creating high quality element meshes can be a simple task. Simply select where to create the elements, what type to create, and with what property and FEMAP will do the rest. Typically, you must define the property before creating the elements, although if no property is specified, FEMAP will prompt you to create one.

5.2.2.3 Element Shape Quality

Once you have created a mesh, always check all elements for distortion with the *Tools, Check, Element Quality* command. You can set maximum distortion criteria and make a group of any distorted elements. Fix all distorted elements, if possible, before adding any loads or constraints. This is especially important if the distorted elements are in a key region of the model.

5.3 Hexahedral Modeling and Meshing

The following section gives an explanation of the steps necessary to perform solid hexahedral meshing. For more information, see *FEMAP Commands*.

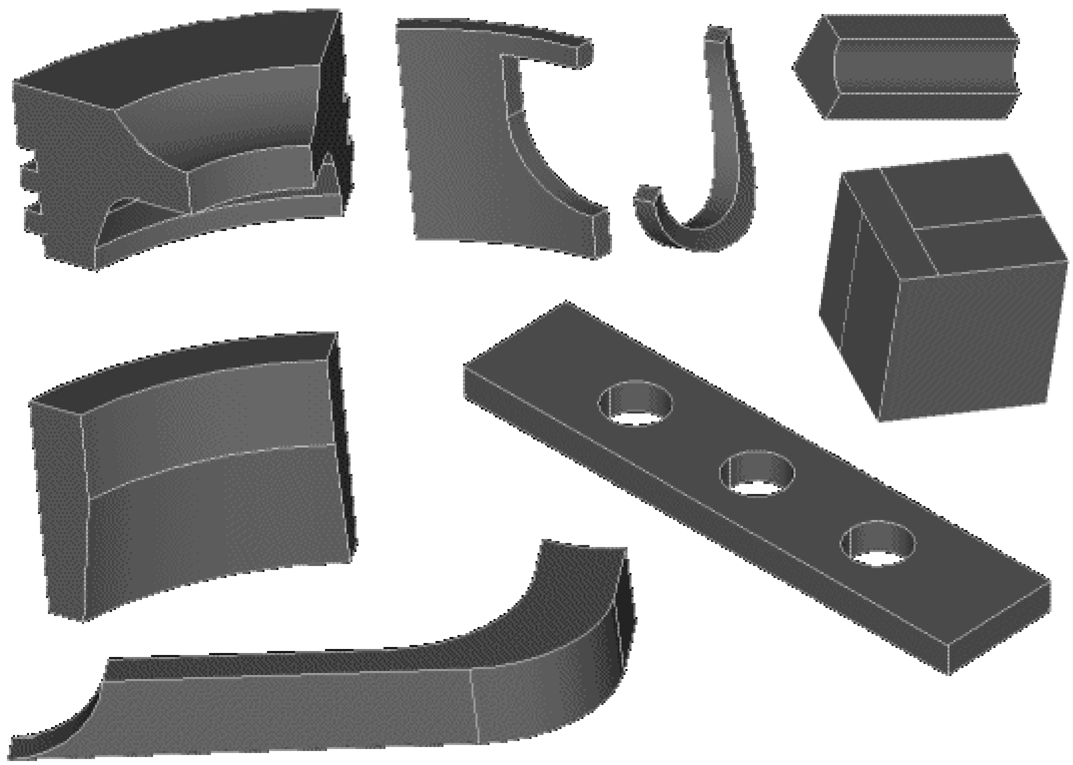
5.3.1 Geometry Preparation

Preparing the geometry is the most critical part of the hex meshing process. Complex solids cannot be automatically hex meshed, but if divided properly into simpler solids, a full or partial hex mesh can easily be generated.

5.3.1.1 Identifying Hex-Meshable Solids

The first step in solid hex meshing is to be able to identify hex-meshable solids or regions within solids. These would include, but are not necessarily limited to, six sided solids, extruded solids and revolved solids. Some examples are shown below.

Examples of solids that can be hex meshed:



You must follow a fairly strict procedure for most solids to crete a hex mesh.

1. Subdivide your model into hex-meshable solids.

2. Set the mesh sizes using *Mesh, Mesh Control, Size on Solid*, with the hex meshing option.

3. Verify that all solids are hex-meshable, and are properly linked to adjacent solids. If not, return to step 1, and continue dividing your solids.

4. Hex mesh using the *Mesh, Geometry, Hex Mesh Solids* command.

Each of these steps is extremely important if you are going to succeed in creating a complete, correct hex mesh.

5.3.1.2 Subdividing the Solid

Once you are familiar with the types of solids that are hex-meshable you must divide your solid into these regions. FEMAP offers a number of commands for this process.

They include the following commands all contained on the *Geometry, Solid* menu (refer to the commands manual for descriptions and use): *Add, Remove, Common, Embed, Slice, Slice Match, Slice Along Face, Embed Face*.

The *Slice* and *Embed* commands are particularly useful when attempting to create hex- meshable regions. If you need to clean up particular surfaces on solids you can use *Geometry, Solid, Explode*. You can then modify these surfaces or create new surfaces with the surface modeling commands. The *Geometry, Solid, Stitch* forms the surfaces back into a solid.

5.3.2 *Mesh Sizing*

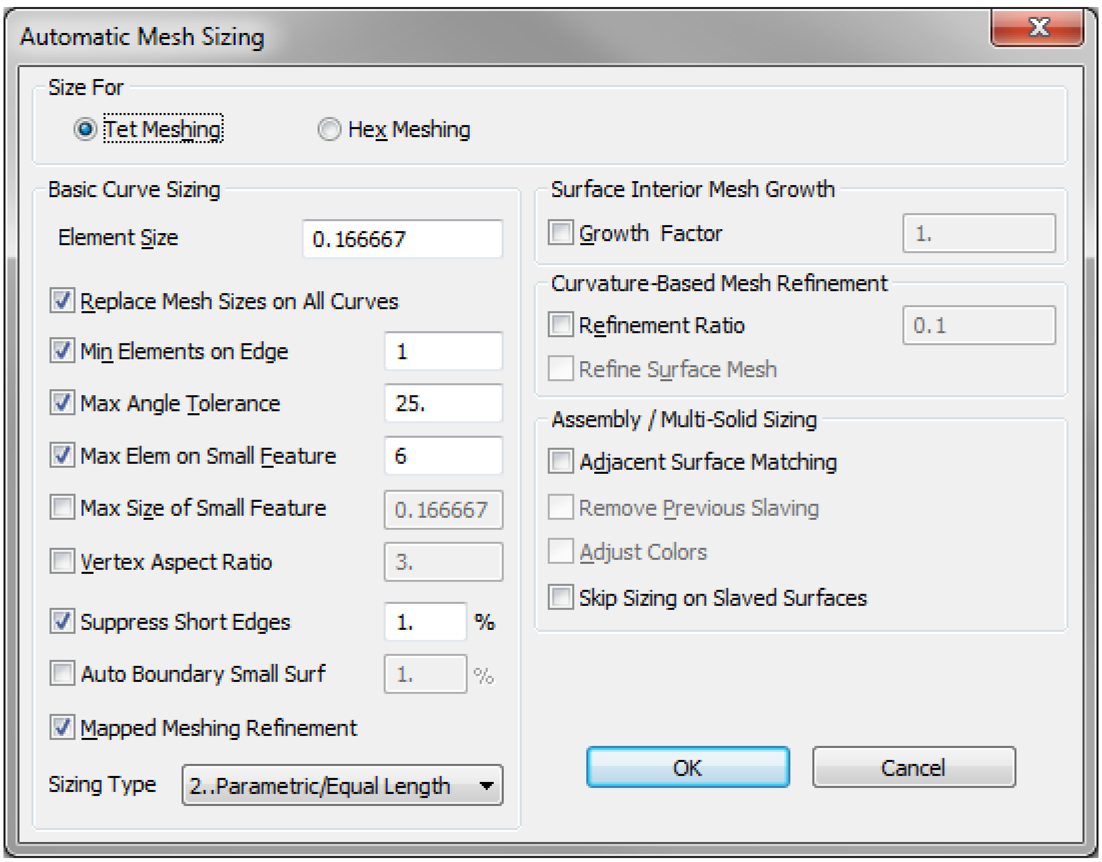
Consistent mesh sizing throughout the model is necessary for hex meshing. It is not possible to transition from a large number of elements to a small number of elements with hexahedral elements. Sizes of elements can change but the number must be consistent. This consistent sizing must propagate through the model, across the multiple solids that you have created. For this reason, local mesh sizing operations have little use in hex meshing. Global sizing and mapped surface approaches and surface linking are much more important.

The *Mesh, Mesh Control, Size on Solid* command (with the *Hex Meshing* option selected) is the primary mechanism to setup the necessary mesh sizing for successful hex meshing. Since many surfaces on your solids must be mapped meshed, curves on opposite sides of those surfaces must have the same number of element divisions. Once you have properly subdivided your part, the *Size on Solid* command handles all sizing automatically. Simply specify a nominal size.

If further mesh grading is required, or you want to modify the sizes that *Size on Solid* has created, you must use great care. If you manually change the mesh size along a curve, you must also manually change the mesh sizes (to the same settings) on all other curves in your solids that must match the first curve to maintain mapped meshable surfaces.

5.3.2.1 *Ensuring Surface Linking*

If you have subdivided your solid surface, linking is required to guarantee a continuous mesh. This is done when you specify the *Hex Meshing* size with *Mesh, Mesh Control, Size on Solid.*



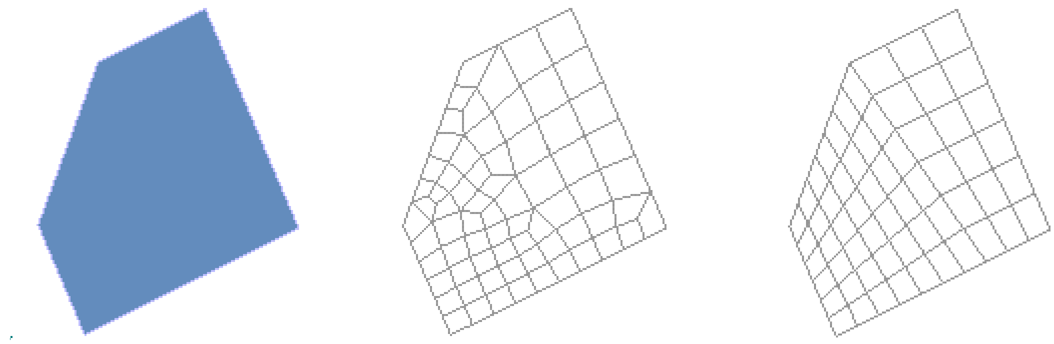
If you select *Hex Meshing* in the *Size For* area, *Adjacent Surface Matching* is checked and grayed. FEMAP automatically looks at all surfaces in all selected solids and finds any coincident ones.

If *Adjust Colors* is checked, you can visualize which surfaces have been linked, and what solids are hex-meshable. *Remove Previous Slaving* will delete any surface approaches.

Note: It is important to remember that FEMAP will only look at the solids you select. If this command is run multiple times on different regions of the subdivided solid, the meshes will not match. To hex mesh the whole part, you must select all subdivisions at the same time.

5.3.2.2 *Specifying Sizes and Surface Approaches*

FEMAP has the ability to use several different meshing approaches on surfaces via the *Mesh, Mesh Control, Approach on Surface* command. These approaches can be used to map mesh surfaces that would otherwise free mesh, and to match surface meshes. These methods are very useful in getting a solid to hex mesh or simply to get a better hex mesh. See Section 5, "Meshing" in *FEMAP Commands* for information on the different approaches that are available.



Free Mesh

Four Corner
Mapped Approach

Specifying individual mesh sizes on curves or surfaces in hex meshing is not recommended. The nature of hex meshing dictates that changes in the number of elements in one area must propagate throughout the model. FEMAP will not automatically update other sizes based on a change. You must do this manually. Be very careful, however, since because you can easily get a discontinuous hex mesh, or no hex mesh at all.

5.3.3 *Hex Meshing*

If you have properly subdivided your solid, and set mesh sizes and surface linking correctly the actual hex meshing is easy. Use *Mesh, Geometry, Hex Mesh Solids* and select the solids. The nodes on the linked surfaces can be automatically merged.

5.3.4 *HexMesh From Elements*

There are times when the hex meshing fails due to bad geometry or adjacent solids with different edge lengths. The command *HexMesh From Elements* provides a way to mesh geometry that has become corrupt thus causing the normal hex meshing technique to fail. This command allows you to individually map mesh the surfaces that make up the solid and then generate solid hexes from the surface mesh.

5.4 *Midsurface Modeling and Meshing*

Midsurfacing is a tool designed for use in certain instances. It is not a general purpose design tool. The solids used must be thin in relation to overall size, sheet metal parts or plastic injection molding parts are good examples.

Although, it is only useful for these type of “thin-walled” parts, it is an extremely valuable modeling tool for these parts. You can produce a much smaller and much more accurate model by meshing midsurfaces formed from a “thin-walled” part, than solid meshing the thin-wall. Differences in model sizes can literally be an order of magnitude in some cases, thereby significantly reducing run time. You even get the added benefit of a more accurate solution in most cases.

The tools for creating midsurfaces are contained on the *Geometry, Midsurface* menu. Their operation is discussed more fully in the *FEMAP Commands*, but a general overview is provided below.

Note: In addition to creating midsurfaces and meshing them, FEMAP provides another capability that can be helpful in creating midsurface meshes on constant thickness parts. In this case, you can simply mesh the outer or inner surfaces of the part, and use the *Modify, Move By, Offset Element* command to move the elements to the midsurface. For more information, see Section 4.8.1.4, "Modify, Move By Menu" in *FEMAP Commands.*

5.4.1 Creating Midsurfaces

The ease of midsurface creation depends greatly on the geometry of your model. Thin flat parts will be nearly completely automatic. Parts with widely varying curvature, small features and/or fillets etc. will take longer and require manual work. Become familiar with FEMAP's midsurfacing capabilities and attempt to prepare your model ahead of time for easy midsurface generation.

5.4.1.1 Automatic

- Fully automatic - the *Geometry, Midsurface, Automatic* command uses a Parasolid face pairing algorithm to create appropriately trimmed and intersected midsurfaces based on pairs of faces. Trying different options available in the command may improve the created midsurfaces.
- Fully automatic (using Pre-V11.1 Midsurface Method) - the *Geometry, Midsurface, Automatic* command has an option that runs the three steps of the midsurface sequence. Be careful when running this command because the delete process may delete surfaces that you need, and they may be hard to recreate.
- Three-step automatic - Generate midsurfaces, Intersect them and then cleanup unnecessary midsurfaces. The *Cleanup* command, when run manually, does not actually delete the surfaces. It places them on a separate layer so you can review them to be sure you want to delete all of them. This approach is often better for very complex solids.

With either of these automatic approaches chances are good you will still need to do some manual cleanup unless the part is a simple thin-walled solid.

5.4.1.2 Manual

Manual midsurface creation and cleanup will involve all facets of geometry modeling in FEMAP. You need to be familiar with all curve, surface, and solid modification tools.

A good general approach for midsurfacing a model is provided below.

1. Perform the three-step automatic process above.
2. Compare midsurface geometry with the solid geometry
3. Manually trim pieces which are not needed to represent the solid.
4. Manually create midsurfaces it the automatic procedure did not produce them. This may require modifying the solid and/or its surfaces.
5. Run *Geometry, Solid, Cleanup* on the midsurfaces to remove internal slices. (This option will require you to then re-intersect all surfaces so nodes match when you mesh).
6. Carefully check your midsurface geometry.

5.4.2 Preparing for Meshing

5.4.2.1 Mesh Attributes

Automatically assigning mesh attributes will create properties of the correct thickness for any midsurface that was generated from a section of constant thickness in your model. The same material will be used for all of these properties, so if you are using different materials edit the properties or assign mesh attributes manually. If you have surfaces that vary in thickness you will have to mesh the surface and then use *Modify, Update Elements, Adjust Plate*. For more information, see *FEMAP Commands*.

5.4.2.2 Mesh Sizing

Mesh sizing is carried out in the same manner as for any plate mesh. Use approaches on surfaces to get mapped meshes. Specify sizes along curves, custom sizes, and use hard points if necessary. Remember to check that mesh sizes match along coincident curves and surfaces.

5.4.3 Meshing

If you have set up your mesh sizes properly and assigned mesh attributes to all of the surfaces, meshing is simple. Select all the surfaces and FEMAP will use their associated attributes. If you have not assigned mesh attributes you will have to mesh surfaces with different properties separately. Once meshing is complete, merge coincident nodes and check your model for free edges. If you have done a good job with the geometry creation there should be no internal free edges, otherwise you will have to fix them. Use the manual meshing commands or go back to the geometry and perform further manipulations.

5.5 Materials and Properties

The following section provide an overview of material and property information required for input in FEMAP. These sections do not provide a comprehensive description of all options of properties and materials. For more information, see Section 4, "Finite Element Modeling" in *FEMAP Commands*.

5.5.1 Materials

FEMAP supports four regular material types and a general tabular data type:

- Isotropic:

  Constant properties in all directions.

  All properties entered as a single value.

- Orthotropic (Both 2-D and 3-D):

  Material properties are direction dependent

  Parameters defined in two planar directions or three principal directions.

- Anisotropic (Both 2-D and 3-D):

  Similar to Orthotropic except more general.

  Specify parameters as a general 3x3 (2-D) or 6x6 (3-D) elasticity matrix.

- Hyperelastic:

  Materials subject to large deformations, such as rubber.

  Input distortional and volumetric deformation or stress/strain data.

  Limited solver support - many solvers do not support this material type - check your solver before using this material type.

- General (Other Types):

  Solver specific types - LS-Dyna, MARC, Abaqus, NX Nastran, and MSC Nastran. Refer to solver documentation for uses and variables

  User defined types - Accessible only in the FEMAP neutral file or through the API.

Isotropic, orthotropic, and anisotropic materials can also have nonlinear material properties associated with them. You set the type of nonlinearity (Linear Elastic, Elastic/Plastic, or Plastic) and input material data such as yield stress and stress-strain curves.

FEMAP also has a library of material types. Although by no means complete, the material library shipped with FEMAP does contain common materials with their respective properties derived from the U.S. Government's MIL-HDBK-5. This library is designed to demonstrate that a material library can be maintained. Many companies prefer to enter their own material properties and structural allowables than accept ones provided from any outside source. Any time you create a new material in FEMAP, and wish to save that material to the library, press the *Save* button in the material creation dialog box.

Note: Through the *File, Preferences, Libraries* command in FEMAP, you can load a different material library than the one that is shipped with FEMAP. This makes it possible for a company with multiple FEMAP users to post a material library on the network that all users can access to obtain the "approved" material properties and allowables. This is also true of all other libraries in FEMAP including the property library.

5.5.2 Properties

Properties are used to define additional analysis information for elements. Most property data is geometric (thickness, area, etc.) but some types will also include inertia, stiffness or mass, as well as other data depending on the type of element/property. There is a direct relationship between the element type and the property type. All elements, except for certain specialized elements like Plot Only or Rigid, must reference a property. Therefore, whenever you want to use a particular element type, you should first create the corresponding property. Similarly, most properties require a reference to a material, so you should create your materials first, and then create properties.

Like materials, a property library exists. You can save any properties you create for future use. Always check Section 8, "Analysis Program Interfaces" to be certain the property will translate correctly into your analysis program.

5.6 Loads and Constraints

It is very important to accurately simulate the real world loads and constraints that are applied to your model. FEMAP provides an extensive array of load types, constraints and methods that make this possible.

5.6.1 Loads

FEMAP provides a wide variety of load types and a wide variety of methods for placing these loads on your FEA model. Loads and constraints are set based, making it possible to categorize them into different cases for different analyses. FEMAP provides four main load categories, with several types of loads under each category, to choose from:

1. Body or global loads:

Acceleration - translational (i.e. gravity), rotational, and varying translational

Velocity - rotational

Thermal - default temperature

2. Nodal loads:

Force/moment

Displacement/Enforced Rotation

Velocity/Rotational Velocity

Acceleration/Rotational Acceleration

Temperature

Heat generation (heat energy/unit volume)

Heat flux (heat energy/unit area)

FEMAP Flow/FEMAP Thermal/FEMAP Advanced Thermal specific loading conditions

3. Elemental loads:

Distributed (load/length across a line element)

Pressure

Temperature

Heat generation (heat energy/unit volume)

Heat flux (heat energy/unit area)

Convection

Radiation

4. Geometry-based loads:

Points

Lines

Surfaces - includes Bearing Load and Torque, which may only be applied via a surface-based load.

All four load categories can be used for static, nonlinear or dynamic analyses.

Body loads are applied to the entire body, therefore only one block of body load information can be specified for each load set. The body loads are most often used to simulate gravity, or to define default temperatures for thermal analyses.

Nodal loads can be applied by both the *Model, Load, Nodal* and *Model, Load, Nodal on Face* commands. With *Model, Load, Nodal*, you directly select the nodes for load application. With *Model, Load, Nodal on Face*, you select a particular face of elements, and nodes on that face will be automatically selected by FEMAP. The *Model, Load, Nodal on Face* command also enables you to select the *Adjacent Faces* method for load application. This method will be discussed further below.

Elemental loads can be distributed, pressure, temperature, heat generation, heat flux, convection or radiation. The distributed loads allows you to define a load/length value for line elements, while pressure defines a load/area for faces of planar elements or volumetric elements. Heat flux, convection, and radiation loads are also applied directly to a face, while temperature and heat generation loads are applied just to the element itself.

Geometry based loads can be either nodal or elemental. You apply the loads to geometry (points, curves, surfaces) and use the geometry to orient the loads. Any nodes or elements that are associated with the geometry will have the loads applied to them appropriately upon export for analysis. You may check how your geometry based loads actually apply to existing nodes and elements using the *Model, Load, Expand* command.

Nodal, elemental and geometry based loads can be time, temperature, or frequency dependent. You must first create the function with *Model, Function*, and choose the appropriate types (*vs. Time, vs. Temperature,* or *vs. Frequency*). You simply need to define the magnitude variation as a function of one of these types, and then reference this function when applying the loads. There will be more on functions, nonlinear and transient analyses later in this manual.

When creating nodal loads on faces or elemental loads, you must supply the face of the element(s). There are four methods available to you in FEMAP:

1. *Face ID*
2. *Near Surface*
3. *Near Coordinates*
4. *Adjacent Faces* - the most powerful method for solid or planar elements.
5. *Model Free Faces*

The *Adjacent Faces* method is the most often-used method. Here you choose just one face, easily done graphically, and then specify a tolerance angle. FEMAP will search all the selected elements for faces that are connected to the face you chose and that are within the specified tolerance from being coplanar with an already selected face. This can be used to easily find all faces on an outer surface of a solid, regardless of the surface shape, or other similar operations. M*odel Free Faces* is similar, but places the load on all “Free Faces”, even on “voids” inside the model.

Geometry-based loads can be oriented in a number of ways depending upon the load and geometry type. Some typical methods are normal to surface, components, along curve, etc. The different methods will be available in the *Create Load* dialog box depending on what is chosen. These methods are also available for orienting nodal loads.

There are also other methods of load creation including *Model, Load, From Output.* This is especially valuable when results such as forces and temperatures are returned to the model. You may convert them to the appropriate load type for further analysis. Nonlinear forces can also be defined which creates forces based upon results from values at other nodes.

The other major command under *Model, Load* are *Heat Transfer Analysis, Dynamic Analysis,* and *Nonlinear Analysis*. These commands control options for heat transfer (steady state and transient), dynamic (transient, frequency response, and random response) and nonlinear (static and transient) analysis types, respectively. When performing any of these analyses, you must first define the appropriate conditions for your load set with these commands. It is also important to note which options are supported by your solver, since FEMAP does not support all these options for the different analysis types. For more information, see Section 4.3, "Creating Loads And Constraints" in *FEMAP Commands*.

5.6.2 Constraints

Like loads, constraints must be created in sets. You can create nodal constraints, geometry based constraints or constraint equations. You can use either the *Model, Constraint, Nodal* command or the *Model, Constraint, Nodal on Face* command to apply constraints to prevent nodes from moving in any of six degrees of freedom (DOF), X, Y, & Z translation, and rotations about the X, Y, & Z axes. The only difference between the two commands is that for *Model, Constraint, Nodal,* you select the nodes directly, and for *Model, Constraint, Nodal on Face,* you select the elements and their faces and FEMAP automatically determines the nodes.

With either of the nodal constraint commands, you may also constrain the DOF in any coordinate system. This enables you to more easily simulate real world conditions, as well as take advantage of symmetry in your model. It would be an extremely difficult modeling task if you had to build all models such that they are constrained only in a global coordinate system.

Geometry-based constraints allow you to select points, curves or surfaces to constrain before or after nodes are on them. Geometry-based constraints have three options, fixed, pinned or no rotations. This command does not allow you as much flexibility as the *Model, Constraint, Nodal* command but is more efficient for large or complex areas with simple boundary conditions.

Constraint equations, unlike constraints, do not fix the DOF to a zero value, but they relate the motion or displacement of different degrees of freedom. You can specify as few as two degrees of freedom or up to a total of 70.

Both load sets and constraint sets may be duplicated (*Model, Load/Constraint, Copy*) or combined with other sets (*Model, Load/Constraint, Combine*).

5.7 Connections and Regions

Connections allow you to create connections between multiple entities in FEMAP. A common type of connection is creating a Connector (contact pair) between two sets of entities. Contact conditions can be used to model interactions between Connection Regions on different parts created from Surfaces, Element Faces, Properties, Curves, and Nodes.

Each Solver FEMAP supports has different options for representing contact conditions and these can be specified by creating a Connection Property. Many solver also support the ability to use contact conditions to Glue (Bond or Tie) parts together. This can very helpful in assembly modeling because it allows you to connect different parts with dissimilar meshes together without having to match nodes and element faces up at the interfaces between the parts.

FEMAP has the ability to create contact conditions automatically using specified parameters or manually between two Connection Regions using a particular Connection Property.

There are basically three steps in creating contact for these programs. They involve three different entity creations:

- Connection Property
- Connection Region
- Connectors

This type of contact is currently supported for NX Nastran, ABAQUS, ANSYS, MARC, LS-DYNA, and NEi/Nastran. In most cases, the solver you are using determines which Connection Property will need to be used to create appropriate contact conditions.

The *Connection Editor* dockable pane can be used to view and modify Connectors using a table control.

In addition to the Connection Regions used for creating Connections, FEMAP has additional regions which are each used for a specific analysis purpose in Nastran.

These four Regions are:

- Fluid Region - Creates the MFLUID Entry used in Nastran to create a region of elements to simulate either a finite volume “internal” fluid (i.e. a fluid in a contained area) or an infinite volume “external” fluid (i.e., ship floating in a body of water).
- NonStructural Mass Region - Creates NSM1, NSML1, and possibly NSMADD entries used in Nastran to apply non-structural mass to specific elements or properties. Non-structural is used to represent the mass of non-structural components such as paint, coatings, wiring, thermal blankets, etc. May be used in conjunction with or as an alternative to specifying non-structural mass on individual properties.
- Bolt Region (NX Nastran Only) - Creates a region of elements where you would like to apply a bolt “preload”. The “preload” is a specified torque which has been translated into an axial load, arising from components in an assembly being bolted together.
- Rotor Region (NX Nastran Only) - Creates a region of nodes which you would like to specify as a “rotor” for Rotor Dynamics in NX Nastran. There are also options to set the rotation axis, damping values, and individual rotor load sets

5.8 Functions

Functions allow you to create general X vs. Y tables of information. They are usually used for time or frequency dependent loads or to attach nonlinear information to material properties. Functions are very specialized in their application in FEMAP. If you are planning on doing any nonlinear or transient analyses, you should review this section. If instead you are planning to concentrate on static, modal, or buckling analyses, you may want to skip this section.

There are many types of functions available. They are listed below with the type of analysis or application for which they are most often used.

- *0..Dimensionless*
- *1..vs. Time* - time dependent loads for transient analysis
- *2..vs. Temperature* - temperature dependent material properties
- *3..vs. Frequency* - frequency dependent loads for frequency response analysis.
- *4..vs. Stress* - stress dependent curves for nonlinear material properties
- *5..Function vs. Temperature* - multiple stress/strain curves as a function of strain rate for nonlinear material properties
- *6..Viscous Damping vs. Frequency* - damping for transient/frequency response analysis
- *7..Critical Damping vs. Frequency* - damping for transient/frequency response analysis
- *8..Q Damping vs. Frequency* - damping for transient/frequency response analysis
- *9..vs. Strain Rate* - yield stress as function of strain rate for nonlinear material properties
- *10..Function vs. Strain Rate* - multiple stress/strain curves as a function of strain rate for nonlinear material properties
- *11..vs. Curve Length* - define load magnitude as a function of curve length.
- *12..vs. Parametric Length* - define load magnitude as a function of parametric length.
- *13..Stress vs. Strain* - stress/strain curve for nonlinear material properties
- *14..Stress vs. Plastic Strain* - stress/strain curve for nonlinear material properties for export to those analysis codes that require input in plastic strain.
- *15..Function vs. Value* - multiple curves associated with a given quantity
- *16..Function vs. Critical Damping* - tables obtained for/from response spectrum analysis
- *17..vs. Angle of Incidence* - used in Advanced Themal Interface
- *18..vs. Direction of Incidence* - used in Advanced Themal Interface
- *19..vs. Temp (TABLEM1 Linear,Linear)* - temperature dependent material properties for Nastran
- *20..vs. Temp (TABLEM1 Log,Linear)* - temperature dependent material properties for Nastran
- *21..vs. Temp (TABLEM1 Linear,Log)* - temperature dependent material properties for Nastran
- *22..vs. Temp (TABLEM1 Log,Log)* - temperature dependent material properties for Nastran

Function types 23-33 are used for output functions created by the *Model, Output, Forced Response* command. Function types 34 and 35 are input functions for Nastran Static Aeroelasticity and/or Aerodynamic Flutter. Finally, Function type 36 is used to specify “Acceleration vs. Location”, which required to create a Varying Translational Acceleration body load.

It is important to identify the proper type for the function you are defining, otherwise it will not be properly used when you attempt to analyze your model.

There are three ways to create data for a function in FEMAP. You can choose single value to enter the X and Y values one at a time. You can use a linear ramp where you pick a starting X and Y value, an ending X and Y value and a delta X. The data points will then be interpreted linearly from the start to end for each delta X. The last is an equation. For an equation, you simply enter the starting and ending values of X and the delta X. Then enter Y as a function of X using the !x variable, e.g. sin(!x). These type of equations can be created easily with the Equation Editor.

Press *Control-E* in a text box to activate the Equation Editor. For more information on the Equation Editor, see Section 4.4.8, "Equation Editor - Ctrl+E".

You can also use the *Paste From Clipboard* and *Copy to Clipboard* commands to transfer data from and to other Windows programs such as Excel. In addition to loading conditions, functions can also be used to define material properties. You may create the functions first or use the *New Function* icon button found in the bottom left hand corner of the *Define Material* dialog box. Choose the *Function Reference* tab and select the function from the drop-down list next to the correct material property.

Again, functions are highly specialized for properly pre-processing nonlinear or transient analyses.

5.9 Groups, Layers and Viewing Your Model

In addition to the numerous pre- and post-processing options provided by FEMAP to make the generation and interpretation of FEA easier, FEMAP also provides a wide array of viewing options that play a key role in increasing your FEA productivity.

The options and methods for controlling how your model is displayed on screen can be divided into two broad categories:

View Select, View Options, and View, Visibility:

View Select controls the top level display options. With *View Select* you can control whether your model is displayed in hidden line or plain wireframe mode, turn on and off stress contours, animations and deformed plots, etc. *View Options* provides the detailed control over how entities are displayed, i.e. what color elements are drawn with, whether or not labels for nodes are displayed, whether or not perspective is turned on, etc. *View Options* also provides extensive control over the display of post-processing options. *View, Visibility* provides a single “tabbed” interface to control the visibility of different entity types and/or labels, groups, layers, loads and constraints. *View, Visibility* also has options to toggle on/off the display of elements based on element type, element shape, and/or associated to materials or properties.

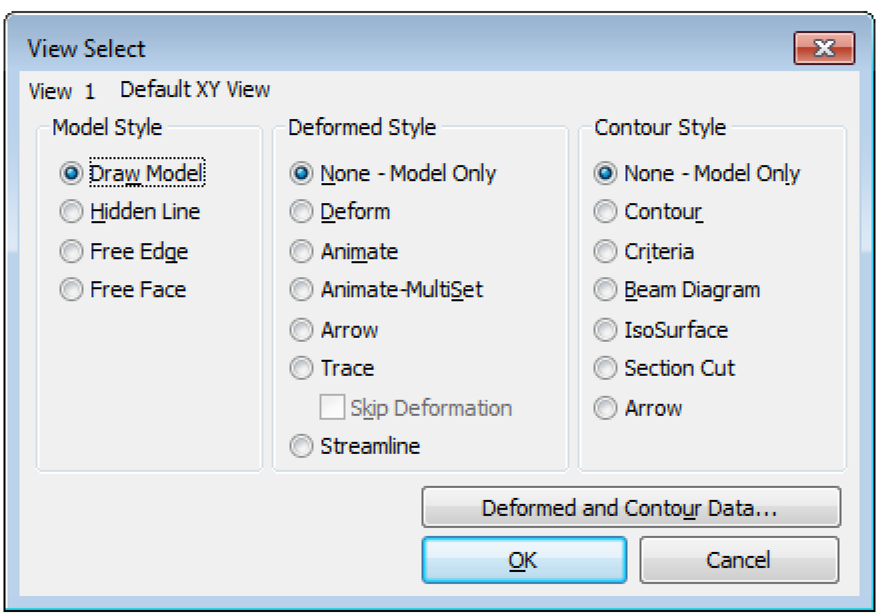
Groups and Layers:

By using groups and layers, you can segment your model into smaller, more manageable, discrete pieces. These pieces can then be used to minimize the amount of information presented in the graphics windows or in printed reports by specifying which group(s) will be seen or which group(s) will be used to create a report. Groups and layers also make it easier to manipulate, update, and apply loads to your model.

5.9.1 Working with View Select and View Options

5.9.1.1 View Select

View Select provides top level control of how your model is displayed.

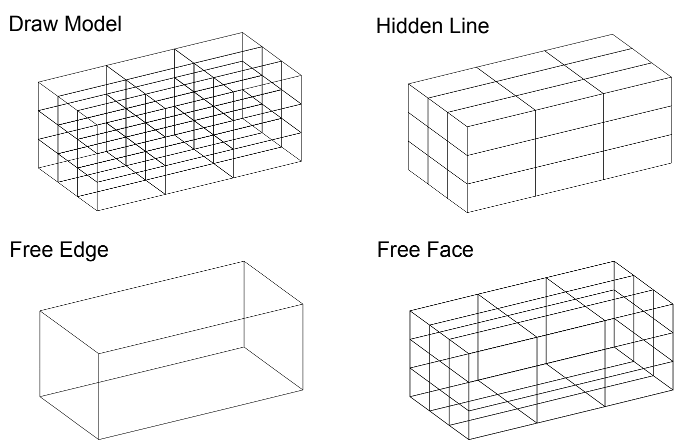


This section describes the *Model Style* options. The *Deformed Style,* and *Contour Style* options are discussed in Section 5.11, "Post-processing".

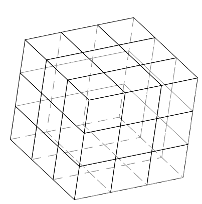
FEMAP provides numerous styles in which you can display your model. Each style provides certain benefits. Choice of the best style depends upon what you need to accomplish. The following table describes all of the styles, their advantages and disadvantages:

Style	Description	Advantages	Typical Usage
Draw Model	Simply displays all entities.	Fast. Everything visible. Usually best “working mode”. Good for screen selection.	Complex 3D models can be hard to visualize. Entities drawn on top of each other may make it difficult to locate a particular detail.
Hidden Line	Sorts all elements, then displays from the back of view. Only shows entities which are visible - hidden lines are removed.	Good for final display and visualization of complex 3D models. Can be helpful for screen selection in complicated models.	Not usually best for picking - many entities are not visible.
Free Edge	Finds and displays all element edges which do not join to another element.	Can quickly point out holes or disconnections in your model.	Usually not used for a working mode. Intended for checking model.
Free Face	Finds and displays all element faces which do not join to another element.	Can quickly point out disconnections between solid elements. Reduces complexity of solid model plots. Can help to find duplicate plate elements.	Usually not used for a working mode. Intended for checking model.

The pictures below show examples of the various model styles.



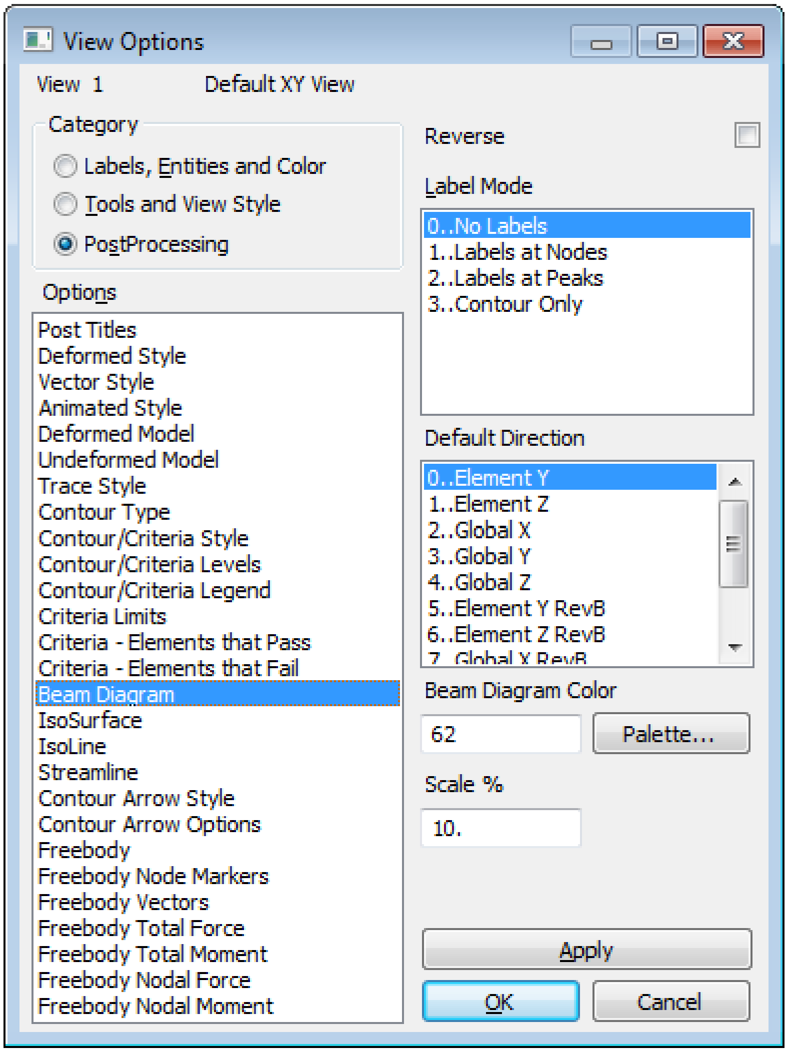
Although the hidden line removal options do require substantial calculations, and are therefore somewhat slower, they can often be the best approach to understanding a complex model. This is especially true for 3D models. After you make the first hidden line display, FEMAP retains a display list of the sorted information. This dramatically speeds up redrawing hidden line views. For more information, see Section 6.3.2.1, "Window, Redraw..." and in *FEMAP Commands*.



For solid element models, you can also use the *Free Face* option to simulate a hidden line view. In fact, you can even use this mode to show hidden lines in a different line style (like dashed), instead of removing them. To remove backfaces, use the *Fill, Backfaces and Hidden* option, under the *View Options* command, and chose one of the “Skip” methods. Choose the *Show All Faces* method to show hidden lines as a different color/style, then go to the *Free Edge and Face* option and set the *Free Edge Color* to *Use View Color*. Finally, choose the color and linestyle that you want to use.

5.9.1.2 View Options

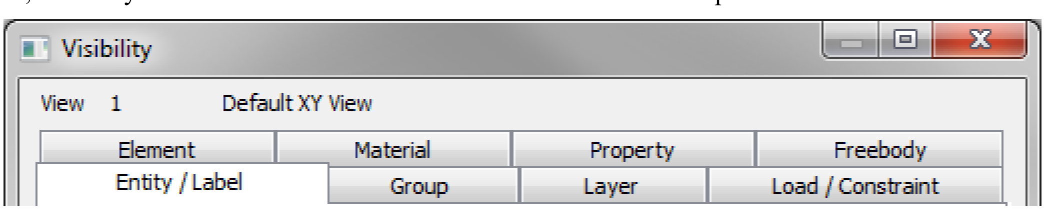
The *View Options* command in FEMAP provides detailed control of the display of all entities in the FEMAP Graphics window(s). Each view in FEMAP is independent, and the *View Select* and *View Options* changes will affect only that view, unless you select the *All Views* option. The quickest method to assess how *View Options* can help you tailor the display of a finite element model is to experiment with the various settings.



As you can see from the *View Options* dialog box, there is an enormous amount of control over how your model is displayed. Describing how each option affects the display of your model is beyond the scope of this manual. For more information, see Section 6.1.5.3, "View, Options..." of the FEMAP *Commands* manual.

5.9.1.3 View Visibility

Controls the visibility of Entity Types and Entity Labels, Groups, Layers, Loads and Constraints, Regions and Connectors, Solid Geometry, Freebody entities, Aero Entities, and Elements using various shape, type, and association criteria (elements referencing Materials or Properties) all in one "tabbed" interface. Please see Section 6.1.4, "View, Visibility..." of the FEMAP *Commands* Manual for a full description.



5.9.2 Groups and Layers Overview

Some main points about groups and layers that will help you understand them better:

- Each entity in FEMAP can have only one layer reference.
- An entity can be in more than one group.
- Any combination of layers from none to all can be displayed at any time.
- A model can have only one “active” group at a time.
- FEMAP graphics windows can use the entities in a group to display one of the following:

  - Entities from the active group.
  - Entities from a single specified group.
  - Entities from a any number of selected groups.
  - Whole Model, i.e., no group.

Groups are designed to mimic how FEA models were numbered and arranged when there were built by hand. For example, in the aircraft industry, a model of a complete aircraft would be very carefully numbered. All the nodes and elements at a frame at a particular location along the fuselage would be numbered in such a way to clearly identify them as belonging to that frame. FEMAP grouping makes it very easy to isolate portions of a finite element model that are numbered in such a manner.

Layers are designed similar to layering in most CAD systems. The name layer comes from the clear sheet of paper analogy for CAD layering, where all the entities associated with a given layer would be drawn on a clear sheet of paper, and only the “active” clear sheets would be overlaid to produce a visual image.

Automatic Generation of Groups

Once you become proficient in FEMAP, you will probably find yourself creating groups as you build a finite element model to keep important areas of the model together for use downstream. If you do not do this, or if you import an existing model, FEMAP has several tools for automatically grouping together portions of your model based on changes in material properties, element properties or even geometric regions.

Combining Grouping, Layers and View Options

Between grouping, layers, the *View, Visibility* command, and the wide array of *View Options*, you have a tremendous amount of control over how your model will be displayed on screen. However, with all these different methods of control you can have problems. These three methods of view controls are not exclusive. They each affect one another.

For example, say you create a new group, add all elements of property 1, and all the nodes associated with these elements, and then use *View, Visibility* command, *Group* tab, to display just that group. You would expect to see the exact entities that you just put in the group. The problem arises out of the fact the layering and the options picked in *View Options* also come into play. If all the nodes that were added into this group were on a layer that is not currently being displayed, they will still not be displayed. Similarly, if the nodes have been turned off in *View Options*, they will not be displayed.

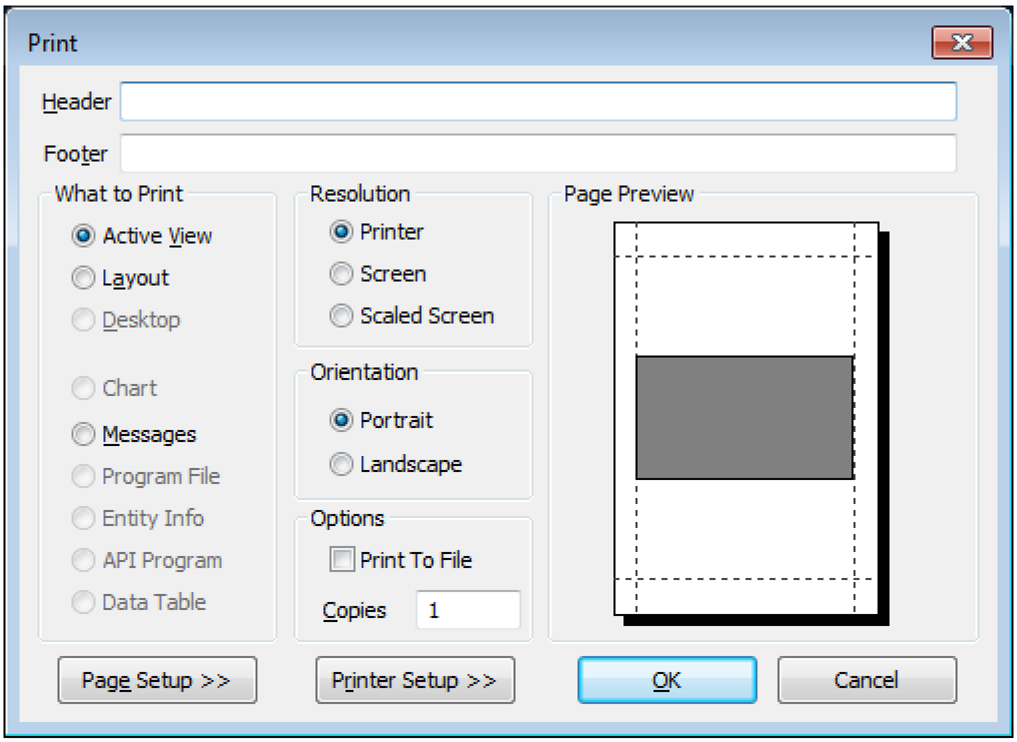
If you ever get into the situation where you think something should be visible on the display and it is not, first check *View Options* and verify that it is on. Next, check *View Layers* and verify that the layer associated with the missing entities is being displayed. Finally, make certain that if a group is being used for the view, that the missing entities are actually in the group. Once you become more familiar with FEMAP and the various options which control the model display, the benefits of the multiple view options methods will become apparent.

5.9.3 Printing

As a Windows application, FEMAP provides What You See is What You Get (WYSIWHG) printing. By default, graphics sent to any printer are vector images, the actual lines, curves and polygons that comprise the graphical representation of your model on screen. As a vector image, the printer driver will break the components down into the colored (or gray scale) dots that form that actual print out. In this manner, FEMAP takes full advantage of the resolution of the output device. Traditional DOS-based FEA (and some Windows ones too), simply dump the bit-map of the screen to the printer. By doing so you are limited to the resolution of the screen, and not that of the printer.

To print any graphics window or text based “dockable pane”, select *File, Print* from the FEMAP menu. If you have more than one graphics windows displayed, you will need to make the window that you want to print from the active graphics window. To do so, simply click the mouse in the window.

File, Print displays the *Print* dialog box, which provides control over how your FEMAP graphics or “dockable pane” (i.e. Chart, Messages, Program File, Entity Info, API Program or Data Table) will be printed.

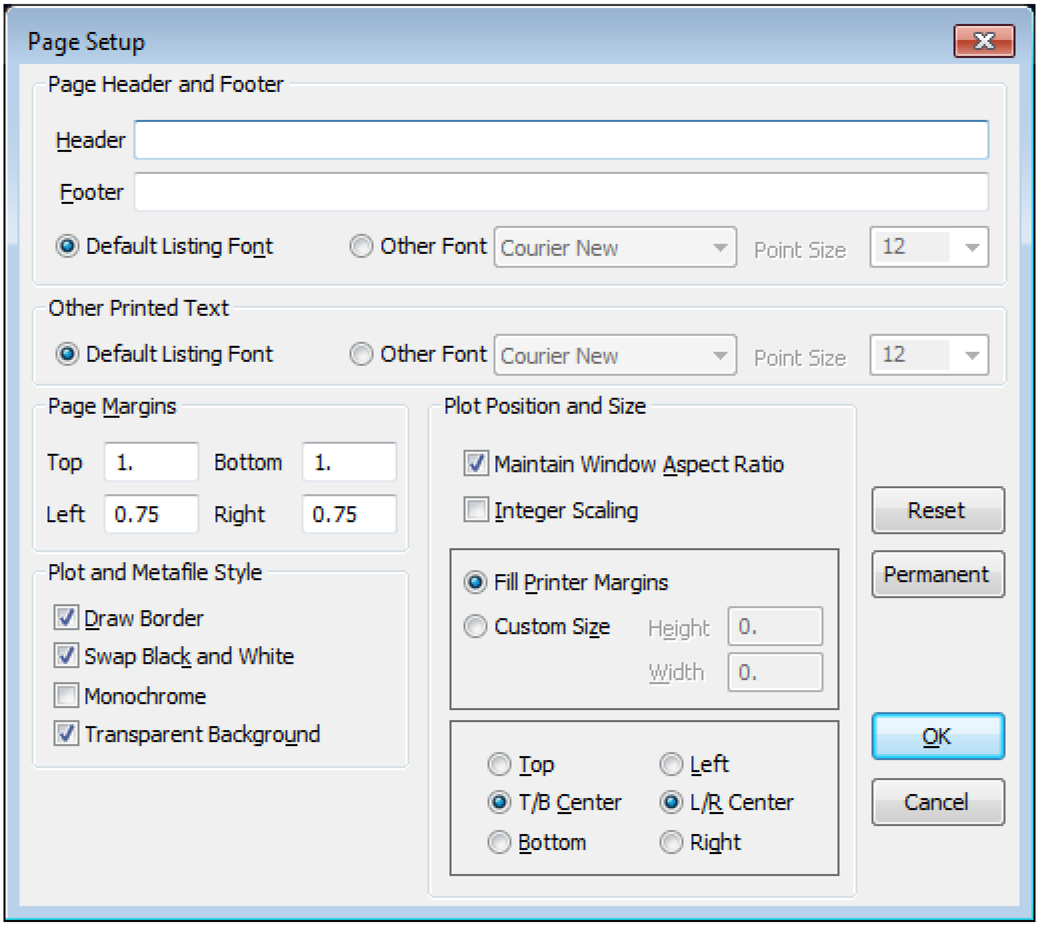


You can quickly add a header and footer to describe the plot being made in more detail, as well as adjust the *Page Setup* and *Printer Setup*.

Printer Setup is most useful for changing the orientation of the plot between landscape and portrait and for controlling aspects particular to your printer.

Page Setup

Page Setup controls aspects more closely related to FEA, including the *Plot and Metafile Style*. Here, you find the *Swap Black and White* very useful if you work in the FEMAP default black and blue shaded background with white elements. Without *Swap Black and White,* any prints made would be "What You See is What You Get", including the black and blue shaded background. With *Swap Black and White*, all black entities are switched to white and vice-versa, which saves you toner and makes the plot easier to see.



5.10 External Superelements Modeling

FEMAP supports Superelements for both NX and MSC Nastran. In general, a particular project and/or model will dictate whether or not Superelements should or must be used for the analysis.

Note: Superelement modeling is an advanced analysis and modeling technique and should really only be done by individuals familiar with using Nastran Superelement technology.

What is a Superelement?

Superelements evolved from the technique known as sub-structuring. It is a method to solve a Finite Element Model (FEM) in a partitioned manner. In other words:

1. Partition the model into parts

2. Reduce/solve each part in terms of its boundary matrices

3. Combine the boundary matrices into what is called the residual

4. Solve for the “residual”/assembly results.

The extra input to NASTRAN (in addition to normal FEM) identifies what nodes and elements belong to which superelement, how the different parts are connected, and then what operations are performed on each part. Basically a superelement is just a collection, subset, or group of nodes, elements, loads, and constraints in an FEM.

Why use Superelements?

Limited Computer Resources - When computers were less powerful, a partitioned solution was needed to allow large problems to be solved one piece at a time. While not as important as it once was, this may still be an issue.

Partial Redesign Solution Efficiency - A partitioned solution allows for partial redesign solution efficiency. If the solution database is saved, only needing to redo the modified superelement could save lots of time.

Model Creation Efficiency - Partitioned input allows different groups of people to work on different parts then a system integrator or integration group can assemble the parts.

Limiting Output to Relevant Data - Partitioned output might be needed or preferred, as it offers the ability to send only relevant results to a particular group.

Security or Confidentiality - With external superelements, only matrix data can be transmitted and system integrator only knows how a part behaves and interacts, not the shape of the part/assembly or other specifications.

Dynamic Solution Efficiency - Allows creation of a much smaller model for dynamic analysis, but one that fully represents the dynamic behavior of all components interacting.

Global – Local Analysis - Allows refinement of a local area which then replaces only a portion of the model.

Facts about Superelements

- Nastran performs a static or Guyan reduction to reduce the superelement to it’s boundary nodes.
- For Statics, the solution is exact. The FEM is divided into superelements by the user in any manner and will produce exactly the same answer as a non-superelement solution.
- For any dynamics solution, solution is not exact, accuracy depends on how boundaries are chosen and what methods are used to supplement the model.(The mass and damping reduction is not exact).
- All solution sequences in Nastran in the 100 range (for instance, SOL 101 for Static Analysis, SOL 103 for Normal Modes Analysis, etc.) are superelement solutions. By default, the entire model is put into superelement 0, also called the residual, and solved as a single superelement problem. This is also called a residual only run.
- The residual or superelement 0 processing and solution is always performed last. All other superelements are called “upstream” superelements.
- Nastran determines superelement membership based on nodes. The user specifies what nodes belong to each superelement, any nodes not listed belong to residual (Superelement 0). Nastran then determines where elements, loads, constraints go based on the nodes.

Superelement Terminology

Types of superelements (SE):

Main Bulk Data SE (Supported by FEMAP) - SE defined by SEID on GRID entry or SESET entry in the normal main bulk data section of the model.

External SE (Supported by FEMAP) - SE saved as boundary node and matrix data only, created in a standalone solution and saved in one of several formats(*.pch, *.op2, *.op4, *.db).

Part SE - SE defined by delimiters in bulk data “BEGIN SUPER” bulk data is partitioned by these delimiters. The input file is now order dependent.

Primary SE (Supported by FEMAP) - SE that references regular bulk data nodes, elements, etc.

Secondary SE - SE that references a *Primary SE* with some transformation added (mirror, rotate, translate)

5.10.1 Creation of an External Superelement using FEMAP

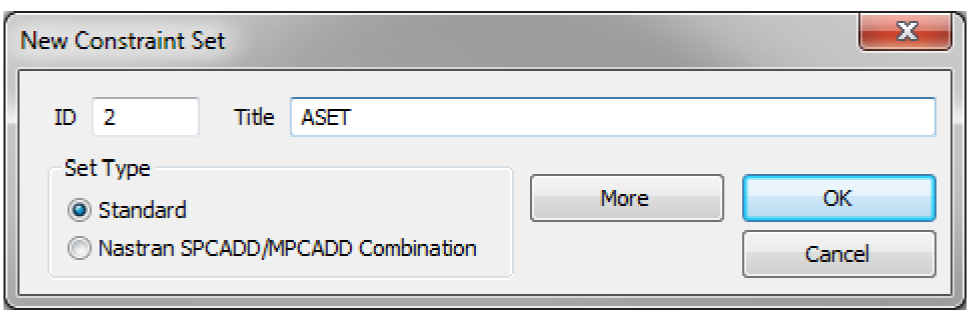
This section takes a step-by-step approach for setting up an External Superelement analysis in FEMAP. There are typically 2 approaches to properly create an External Superelement model. One is to only have nodes and elements from the portion of the overall structure in the model file. The other is to select a group from the “Portion of Model to Write” drop-down list in the *NASTRAN Bulk Data Options* dialog box found in the *Analysis Set Manage*r to limit what is written to the Nastran input file. The “External Superelement Creation” run does nor require a Superelement license for Nastran

Note: If using the group approach, it is probably best to first add nodes or elements into the group, then use the “Group, Operations, Add Related Entities” command to include any materials, properties, loads, and boundary conditions associated with the nodes or elements already in the group. Turning on “Group, Operations, Automatic Add” may also be a good idea.

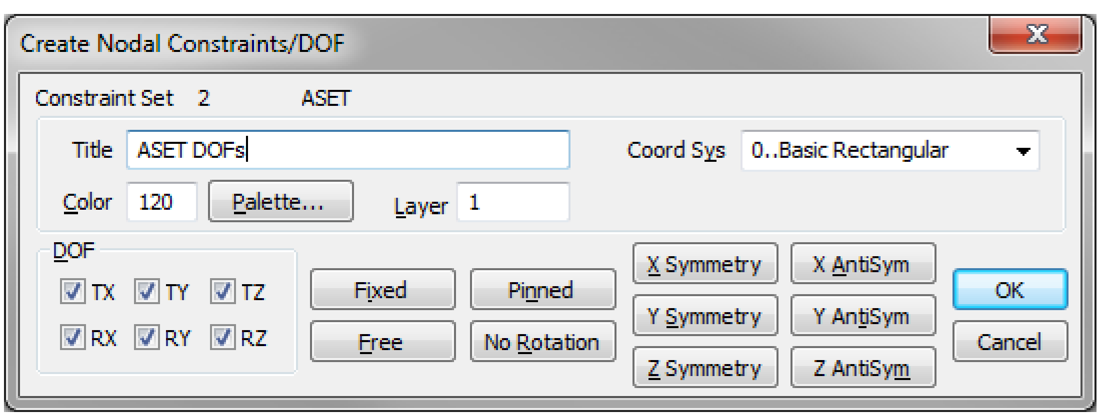
Specify a Nastran ASET using a *FEMAP Constraint Set*

For a basic model, Nastran requires the definition of the physical components to be retained by the reduction process. This is accomplished with ASET Bulk Data entries. The nodes constrained in the ASET are known as the “boundary nodes” of the Superelement.

To create the ASET, create a normal constraint set using *Model, Constraint, Create/Manage Set* and optionally set the title to “ASET”.



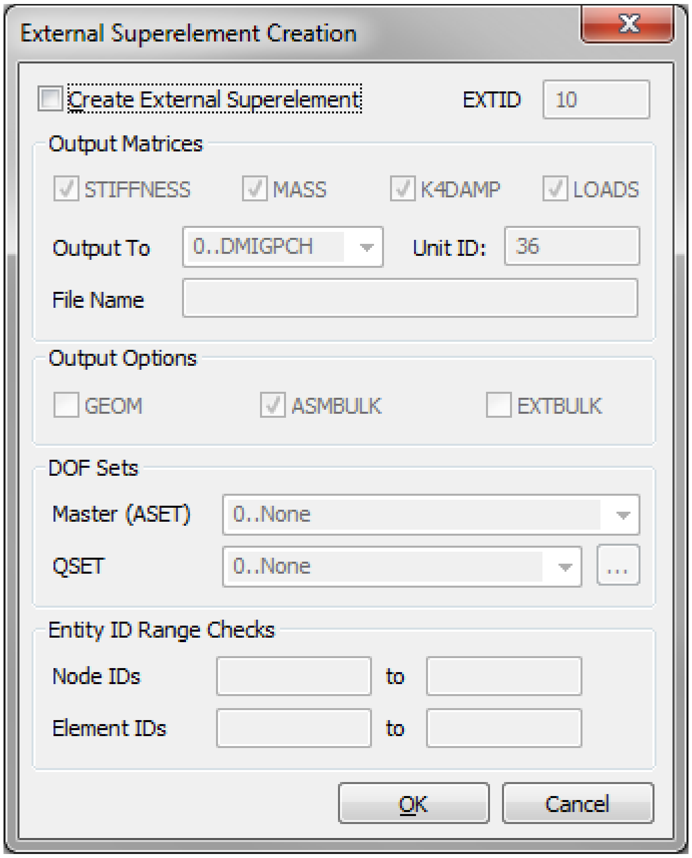
Now, apply “fixed” constraints to the desired nodes using *Model, Constraint, Nodal*, to specify the ASET DOFs.



Selection of the ASET nodes is very important. They are really the only nodes that can transfer motion to the rest of the model from the Superelement. Any nodes which represent a connection to the residual structure should be included. Also, enough nodes should be selected to insure all of the required modes are recovered from the Guyan reduction. Finally, it may be useful to include any nodes which have nodal loads applied in the model.

Create Analysis Set and Specify Options for *External Superelement Creation* Run

Create a Normal Modes analysis using NX Nastran as the solver in the *Analysis Set Manager*. Click *OK* to return to the *Analysis Set Manager*, then expand the *Master Requests and Conditions* branch, highlight *External Superelement Creation*, then click *Edit*. The *External Superelement Creation* dialog will be displayed:



Step 1: Check the *Create External Superelement* check box to make the other options in the dialog box available. These options will be used to create the EXESEOUT Case Control for Nastran. Also, use the *EXTID* filed to specify an ID for the External Superelement.

Step 2: Select the desired output matrices and output format using the *Output To* drop-down. By default, the STIFFNESS, MASS, K4DAMP, and LOADS output matrices will all be included in the output file. When either "1..DMIGOP2" or "2..MATOP4" are used, a *File Name* must also be entered. The appropriate .op2 or .op4 extension will be supplied if not included in the *File Name*. A *UNIT ID* must also be specified and is needed when setting up the "Assembly" run.

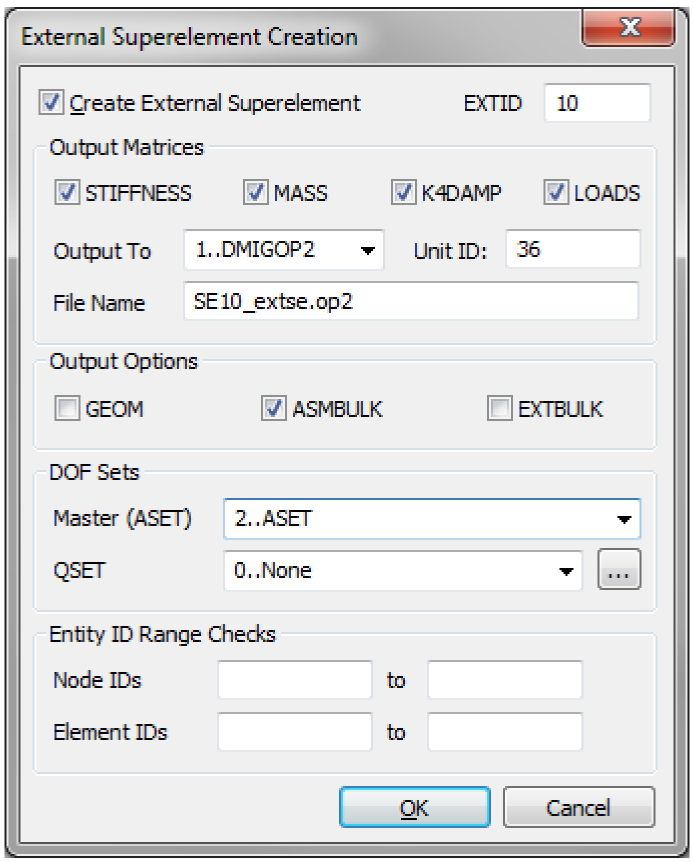
Step 3: Select *Output Options*. Creates data blocks in the selected output file format. *ASMBULK* is on by default, and any combination of *GEOM*, *ASMBULK*, and *EXTBULK* may be selected. This section is not available when *Output To* is set to "0..DMIGPCH", as *EXTBULK* is ignored and the other data blocks are always included.

Step 4: Select the ASET constraint set using the *Master (ASET)* drop-down. Optionally, specify a QSET using the *QSET* drop-down.

Note: If Craig-Bampton modes are needed, a QSET must also be selected. To easily create a QSET, click the "..." button next to the *QSET* drop-down. In the Create SPOINTS dialog, enter the *Number of SPOINTs* (each represents a Craig-Bampton mode) and the *Start ID* for the SPOINTs.
If using the group approach, any SPOINTs and constraints created by the *Create SPOINTs* dialog box will automatically be added to the group specified in "Portion of Model to Write", regardless of the setting of "Group, Operations, Automatic Add". If the group specified in "Portion of Model to Write" is changed, it may be required to use the *Create SPOINTs* dialog box again if the original SPOINTs and constraints have not been manually added to the newly selected group.

Step 5 (Optional): Enter an upper and lower bound of *Node IDs* and/or *Element IDs* in the *Entity ID Range Checks* section to make sure there is no overlap between different Superelements. This is not required by the External Superelement Creation run, but may be a good check in some instances.

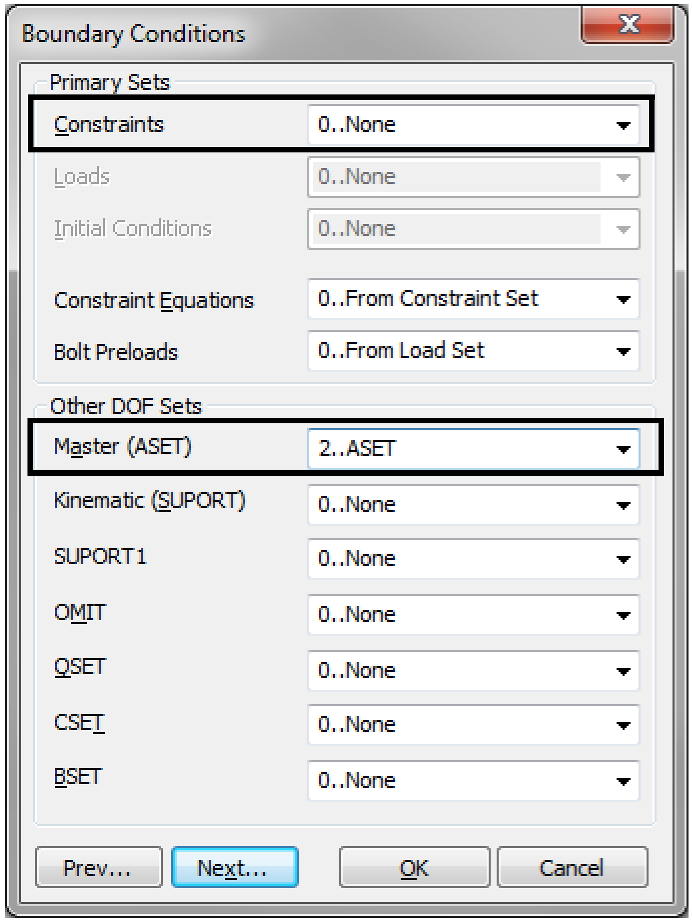
For this example, all the *Output Matrices* are included, "1..DMIGOP2" has been selected, and a *File Name* of "SE10_extse.op2" provided. Only *ASMBULK* is checked in *Output Options* and the ASET constraint set was already selected in the *Boundary Conditions* dialog box, thus is automatically selected in this dialog box.



This concludes the setup of the EXTSEOUT entry for the "External Superelement Creation" Nastran run.

Review Boundary Conditions

In the *Master Requests and Conditions* branch, highlight *Boundary Conditions*, and click *Edit*. In the *Boundary Conditions* dialog box, set *Constraints* to “0..None”, then check to be sure the ASET constraint set is selected in the *Master (ASET)* drop-down, then click *OK*.

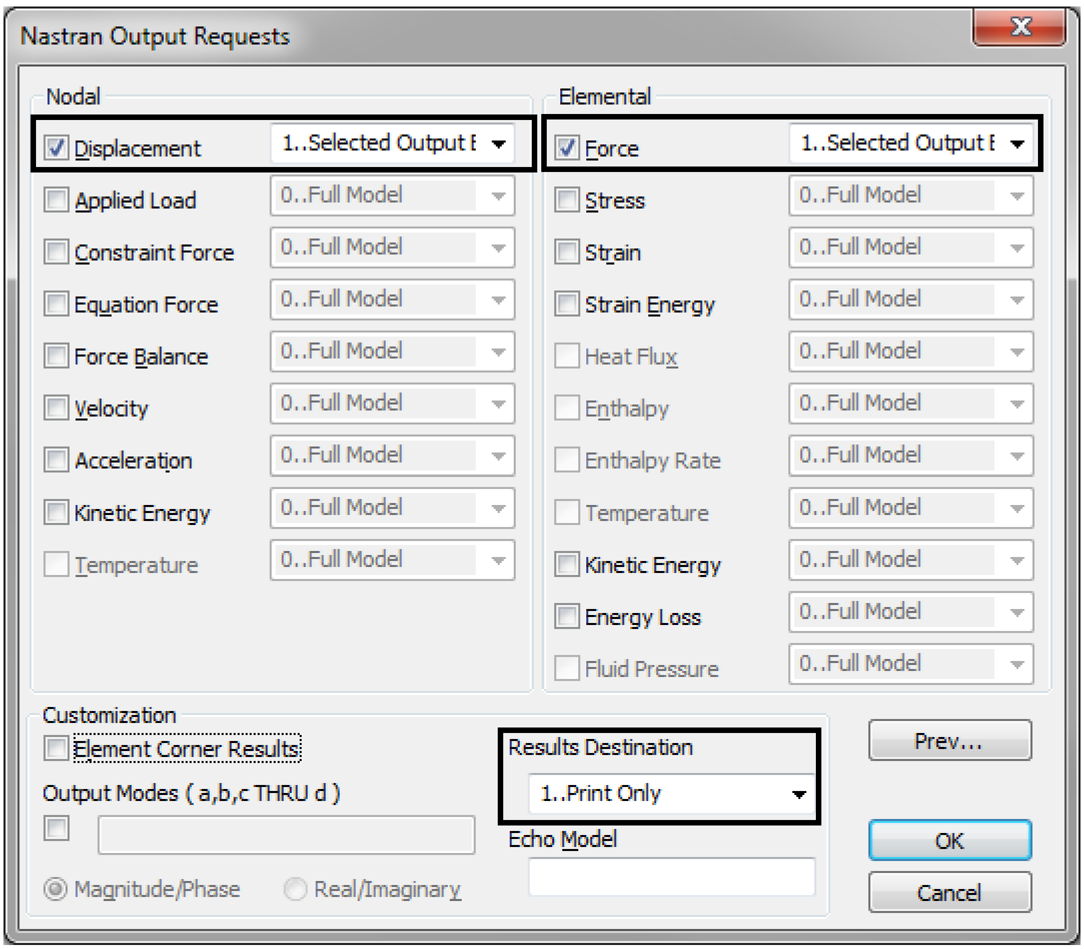


Setting up appropriate Output Requests

The *.pch, *.op2, or *.op4 file created by the "External Superelement Creation" run will include the Output Matrices selected in the *External Superelement Creation* dialog box and "Output Transformation Matrices" (OTMs) for all output requested in the Case Control. The OTMs can make the output file VERY large, so requested output should be limited to only what is absolutely needed in the final "assembly" run.

Also, the "External Superelement Creation" run will automatically create the required *.op2 file. Because of this, there is no reason to have the *.op2 file created by the PARAM, POST, -1 entry, which is the default for a new Analysis Set in FEMAP.

To do this in FEMAP, groups containing only nodes and/or elements of interest should be created. In the *Analysis Set Manager*, highlight *Output Requests* in the *Master Requests and Conditions* section, then click *Edit*. In the *Nastran Output Requests* dialog box, turn on only the desired output types, then select *Groups* using the drop-down list next to each output type. Also, set the *Results Destination* option to "1..Print Only" to have Nastran only produce the printed output file (*.f06) for possible later examination.



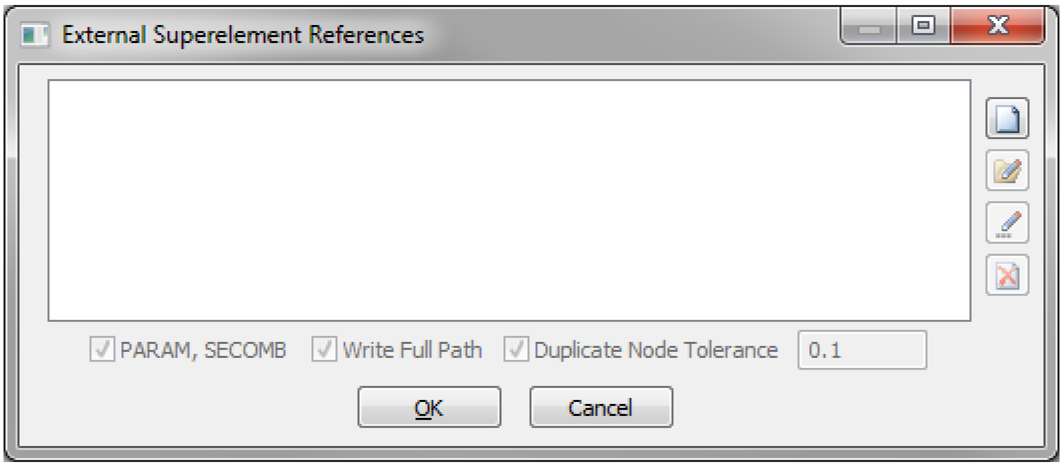
The "External Superelement Creation" run is now ready to be sent to Nastran.

5.10.2 Referencing an External Superelement using FEMAP

Once “External Superelements” have been created, they can now be “referenced” by an *Analysis Set* for use in an “Assembly” analysis run. In the *Analysis Set Manager*, create a new *Analysis Set* with the desired analysis type. Expand the *Options* branch, highlight *External Superelement Reference*, then click *Edit*. Running the “Assembly” in Nastran requires a Superelement license.

Note: If the “Create External Superelement” option is on in the *External Superelement Creation* dialog box, then *External Superelement Reference* will not appear in the *Options* branch.

The *External Superelement References* dialog box provides tools for selecting “External Superelements”:



This dialog box has for icon buttons on the right hand side. From top to bottom, there are:

New - Creates a new “External Superelement Reference” beginning with selection of the output file (*.pch, *.op2, or *.op4), followed by setting the *Unit ID* (must be the same Fortran Unit specified during “External Superelement Creation”), the *Type*, and which *Output Matrices* to use (*.pch file only). In addition, when using either a *.op2 or *.op4 file, a *.asm file corresponding with the *.op2 or *.op4 file must be selected to complete the reference.

Edit All - Used to edit the selected *External Superelement Reference*, starting with selection of the output file.

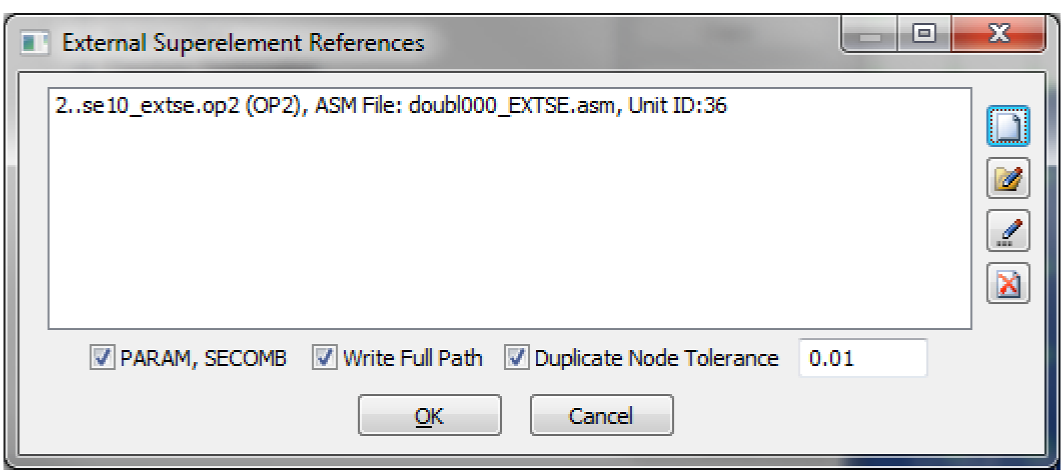
Edit Options - Used to potentially edit the selected *External Superelement Reference*, starting with *Unit ID*, *Type*, and *Matrices*, then possibly the selected *.asm file.

Delete Reference - Simply deletes the selected *External Superelement Reference*.

Each *External Superelement Reference* has a unique ID and any number can be placed into the list.

Across the bottom of the list, options to toggle on/off *PARAM, SECOMB*, *Write Full Path*, and *Duplicate Node Tolerance* exist. A value may also be entered for *Duplicate Node Tolerance* if the default is not appropriate.

A reference shown using a *.op2 file, a selected *.asm file, and a specified Unit ID:



Once all references have been entered, the “Assembly” run is now ready to be sent to Nastran.

5.11 Post-processing

The first step in post-processing is to obtain the results. If your analysis program does not launch from FEMAP and automatically return the results, you must import them. Use *File, Import, Analysis Results* and select the proper format. Select the results file for your model from the standard file selection box using the default file extension for your analysis program.

Similar to loads and constraints, output data is also stored in sets. If you run your model with several different loading conditions or through several different analysis types, FEMAP will keep the output data from each analysis, each mode shape, or each time step in a different output set. Post-processing can be divided into two main categories, graphical and report. Graphical post-processing can be further divided into:

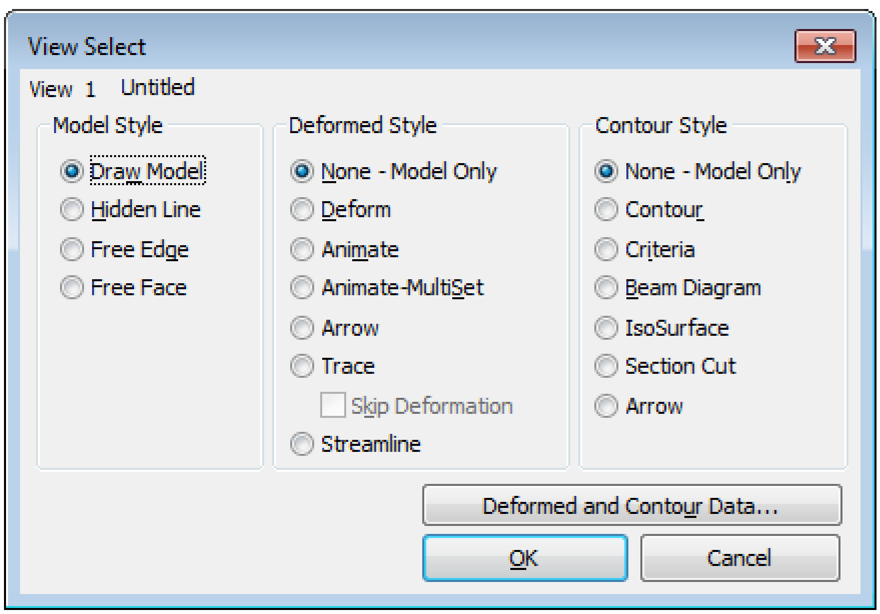
1. Deformation plots
2. Contour/criteria plots
3. Free body plots
4. XY plots

Deformation and contour/criteria plots can be combined in the same view. All model style options (such as *Hidden Line*) are available for deformed and contour styles. Free body plots can be shown in any view with either a deformed and/or contour plot on or off.

Report based post-processing is fairly straight forward, providing text output of results data in a variety of formats, printing options, and sorting options.

5.11.1 Deformed and Contour Plots

The first step in post-processing is to define the type of plot desired, and the data to be used in the display. The *View Select* command is the main control for how your model is displayed, including what post-processing options are being used.



From *View Select* you can invoke five different types of deformed style plots:

- *Deform* - Show a plot of the deformed shape.
- *Animate* - Animate the deformed shape.
- *Animate Multi-Set* - Perform animation across several sets. Good for transient, nonlinear and frequency response analyses.
- *Arrow* - Show arrows representing direction and magnitude of output.
- *Trace* - Similar to *Animate Multi-Set,* except displays trace lines connecting historical positions of nodes.
- *Streamline* - View results from flow analysis (i.e., CFD) using streamlines

For multi-set animation and trace plots, you may also decide to only animate the contours by selecting the *Skip Deformation* option. This can be extremely useful for heat transfer and similar types of analyses.

From *View Select* you can invoke six general contour style options:

- *Contour* - Provides smooth representation of data.
- *Criteria* - Elemental values displayed at centroid of element.
- *Beam Diagram* - Similar to 3-D shear and bending moment diagrams. Display results along the length of line elements.
- *IsoSurface* - Provides interior surfaces of constant values in solid models.
- *Section Cut* - Shows contours through any planar cut of a solid model.
- *Arrow* - Arrows at centroids of elements or on nodes.

Specialized Post-processing Features

The following are some specialized post-processing features in FEMAP.

- *View, Select - Deformed and Contour Data - Section Cut - Multiple Sections:* Works in undeformed, deformed, or animate contour plot modes, but only with solid elements. This allows you to choose up to three independently oriented cutting planes. The location of these planes can be controlled by the *View, Advanced Post, Dynamic Cutting Plane* dialog box.
- *View, Advanced Post, Dynamic Cutting Plane:* Works in undeformed or deformed contour plot mode only, and only with solid elements. Allows you to choose an arbitrary cutting plane and dynamically pass it through a solid model. The value associated with the plane is the distance from the global origin to the plane along the normal vector of the plane. Colors indicate the value associated with the corresponding color on the contour legend.
- *View, Advanced Post, Dynamic IsoSurface:* Works in undeformed or deformed contour plot mode only, and only with solid elements. Allows you to dynamically change the value of the isosurface being shown. The value is from the current output set and vector chosen as the contour vector. The color of the isosurface is controlled in the view options post-processing category. If contour deformed is chosen, the vector for the deformed data is contoured across the isosurface. Otherwise it is a single color chosen from the palette.
- *View, Advanced Post, Beam Cross Section:* Works on beam elements with cross-sections only. Uses results typically recovered from a beam analysis to calculate one of 7 available types of stress, then shows the calculated stress output on the cross-section of the beam. Many options are available to modify the display.
- Dynamic rotation of animations: You can dynamically rotate during animation. However, the animation will pause until dynamic rotation is finished.

Selecting the Data to use for Post-Processing

Control over what data is used in deformed or contour plots is provided by the *Select PostProcessing Data* dialog box. This dialog box is accessed through the *View, Select* command or the Quick Access Menu (right mouse) menu as *Post Data*. It allows you to control the output set and output vectors shown with the deformed and contour plots.

To choose what data is used in the display, choose the output set (A in figure), the data vector to use for deformation (B), and the data vector to use for contouring (C). You can limit the category and type of output you see in the drop-down lists with the Output Set and Output Vector filters. If you are animating multiple sets, you can choose the *Final Output Set* and the *Output Set Increment* to animate as well.

The *Transform* buttons are used to transform nodal output into another coordinate systems of choice or into each node's output coordinate system and/or to transform elemental output into each element's material direction, into another coordinate system, or into a specified vector.

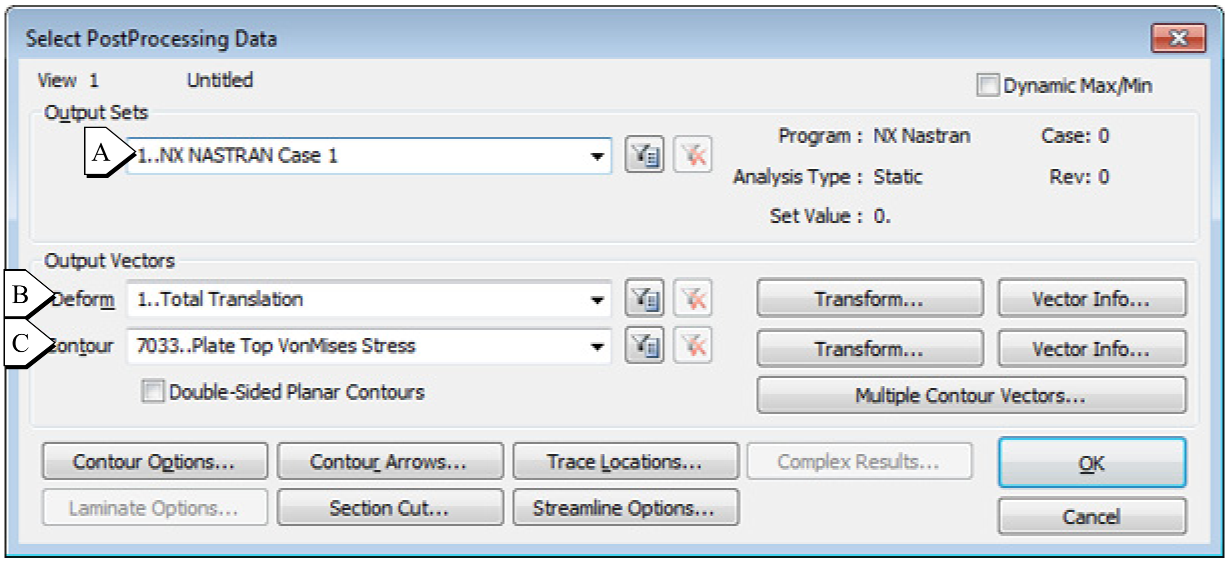
Vector Info displays the *Output Vector Info* dialog box, which provides Max/Min values and the node/element ID where these values occur. In addition, you can also choose to see the values for *Component/Corner Vectors*, if available for the selected vector and other *Vector Statistics*, such as *Sum*, *Number of Entries*, and *Average* value.

Note: For dynamically changing Max/Min values in this dialog box without having to go into *Vector Info*, turn on *Dynamic Max/Min*. Please be aware there may be some delay when changing output sets or output vectors while the Max/Min values are calculated and displayed, especially in larger models.

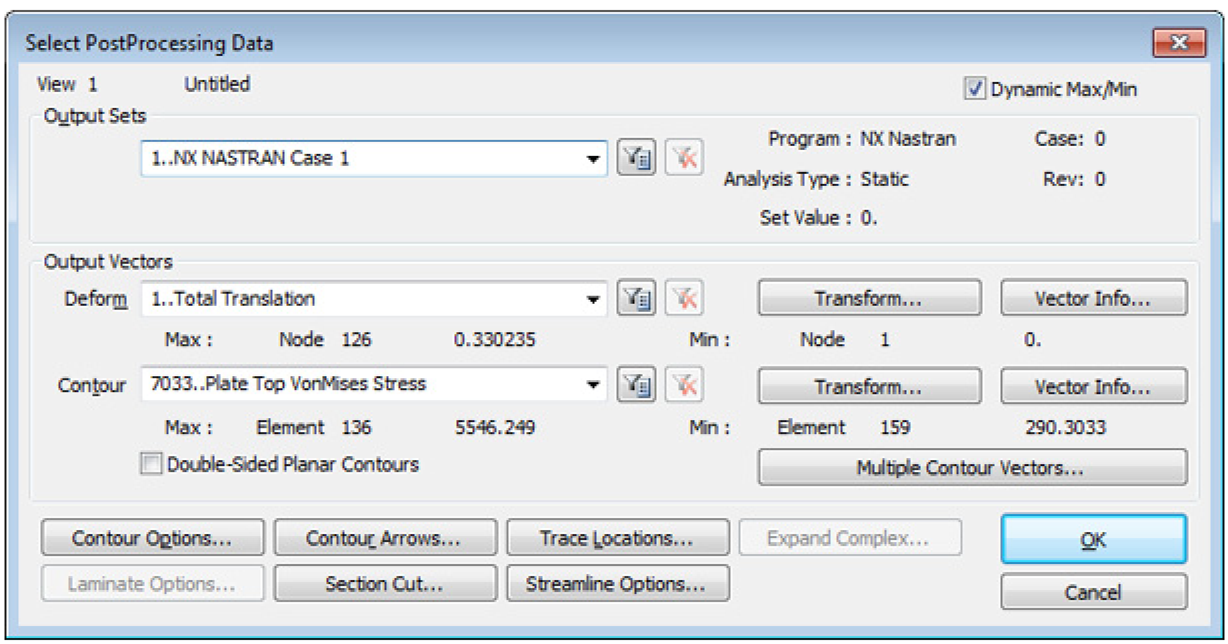
The *Multiple Contour Vectors* button allows selection of up to two additional output vectors for display at the same time as the output vector selected using the *Contour* drop-down (i.e., 3 total). The *Double-Sided Planar Contours* option can be used to automatically show the "bottom" vector when the "top" vector is selected (or vice versa) on the "top" and "bottom" of planar elements.

The *Section Cut* button allows you to select options for Section Cut display. In the *Section Cut Options* dialog box, click the *Section* button to define a "cutting plane" with the standard plane definition dialog box. If you have the *Multiple Sections* option on, then you may define up to three different cutting planes using the *Section #* buttons.

Other buttons exist for choosing *Contour Vectors*, creating *Trace Locations*, setting up display of *Laminate* results, and selection of *Streamline Options.*



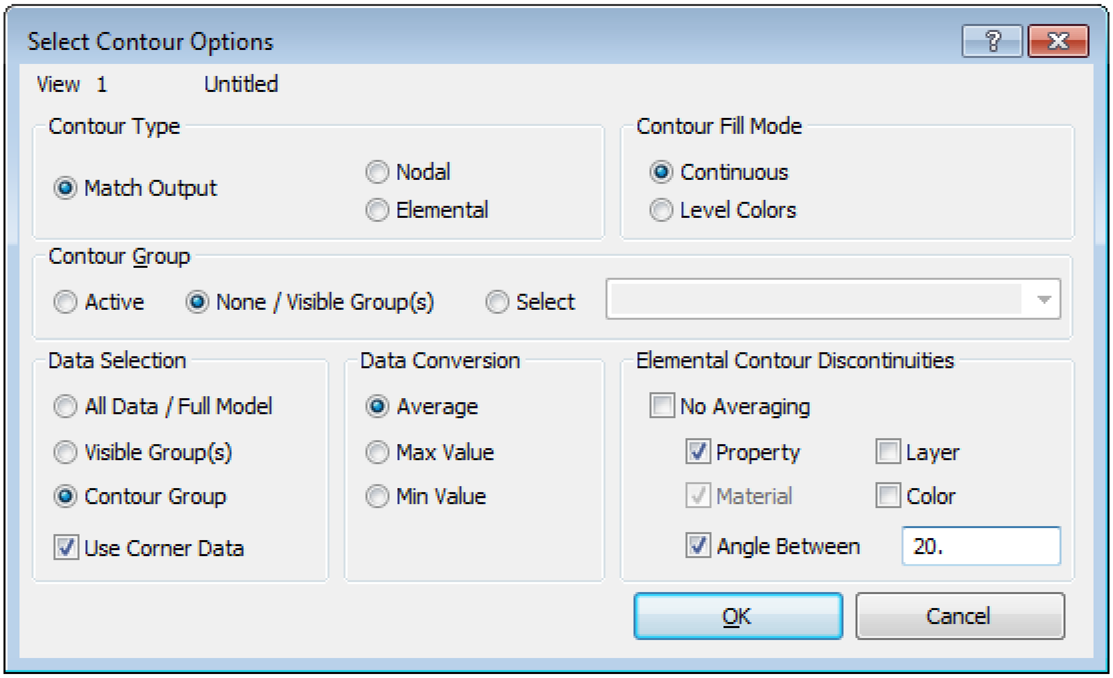
Select PostProcessing Data dialog box showing *Dynamic Max/Min* option turned on:



Contour Options

The *Contour Options* dialog box allows access to the type of contour and data conversion to perform. When you select this option, The *Select Contour Options* dialog box appears.

These options are very important to understand since they control the type of contour and how the data is converted from pure discrete numbers to a visual representation. Improper selection of contour type or data conversion can lead to erroneous interpretation of the results.



This dialog box is separated into six major sections:

- *Contour Type*
- *Contour Fill Mode*
- *Contour Group*
- *Data Selection*
- *Data Conversion*
- *Element Contour Discontinuities*

Each of these areas are discussed more fully below. All of these options can also be accessed through the *View Options* command (*Category - PostProcessing, Option - Contour Type*).

Contour Type

This section allows you to pick from either nodal or elemental contouring. Nodal contouring simply averages all values at the nodes and cannot account for any discontinuities in material or geometry. When *Nodal* is selected, a relatively smooth contour will appear, although the results will not be accurate at material boundaries or property breaks. In addition, the *Other Options* section will not be available. Nodal contouring should not be used across material boundaries or changes in properties such as plate thickness since averaging stresses across these areas results in inaccurate results at the interface.

If elemental contouring is chosen, you can specify which discontinuities in the model to use in the contouring to obtain an accurate representation of the results. This type of contouring is very useful for multiple material models as well as models with plates with that intersect at large angles or have varying thickness. Stresses will not be averaged across these values. The resulting graphics may not be as “smooth” as nodal contouring, especially at material breaks, but it provides a more accurate representation of the results when discontinuities exist in the model. In addition, element contouring allows you to view both top and bottom stresses of plates on one plot, as well as up to two additional output vectors..

Note: Element contouring has the additional feature that if you select *No Averaging* under *Element Contour Discontinuities*, the pure data at the element centroid and corners is plotted without any manipulation. This provides a graphical representation of the pure data. For more information, see "Elemental Contour Discontinuities".

Contour Fill Mode

This section allows you to choose between *Continuous Colors* and *Color Levels* for *Contour Fill Mode*.

Contour Group

You can choose to contour a group while showing the rest of the model with no contours. By default, this option is set to *None/Visible Group(s)*, which simply shows a contour on the entire model or the “visible group(s)”. You can contour the *Active* group or choose a group from the drop-down list next to *Select*.

Note: When using a *Contour Group*, the *Contour Type* will automatically be set to “Elemental” and can not be changed as long as a *Contour Group* is being used.

Data Selection

This section allows you to choose which output data is used to determine the Max/Min values on the “Contour Legend”. The *All Data/Full Model* option will use data for all nodes or elements in the entire model. *Visible Group(s)* will only use the output data from the “visible” group(s) to determine the Max/Min values, while *Contour Group* will use the single group specified in the *Contour Group* section.

In addition, the *Use Corner Data* option allows you to choose if you would like to use any elemental corner data (if it has been recovered from the analysis program) or to skip it for any of these methods.

Data Conversion

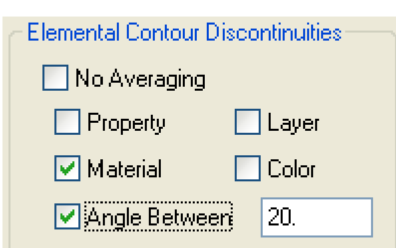
This section controls how FEMAP converts the results from pure data at element centroids, corners, and nodes to the actual continuous graphical representation. There are three options to convert the data: *Average, Max Value*, and *Min Value*.

If *Average* is on, FEMAP will take an average of the surrounding values to obtain a result, whereas *Max* or *Min Value* will just use the max or min value, respectively, of the pertinent surrounding locations. The *Min Value* option should only be used when performing contours for vectors where the minimum values are actually the worst case, such as safety factor or large compressive stresses. You can also choose to use any elemental corner data (if it has been recovered from the analysis program) or to skip it for any of these methods.

The easiest way to understand the data conversion process is through an example. If an interior node of a continuous mesh (no geometric or material breaks and averaging is on) is attached to four elements, there will be four values associated with it for a given stress vector (either corner data or if *Use Corner Data* is off elemental centroidal data). If these values are 100, 200, 300 and 400, an Average conversion would result in 250 at that node, a Max conversion with 400, and a Min conversion of 100. This procedure would be used at all nodal locations to get the basis of the plot, and then FEMAP would produce the corresponding colors between locations. Thus, the data conversion can significantly affect the results if there is a large gradient across adjacent elements.

Hint: You can use the difference in Max, Min and average results to make a quick estimate of the fidelity of the model. If there is a large difference between these two contours, especially at locations that do not have sharp corners or breaks in the model, your FEA model may require a finer mesh.

Elemental Contour Discontinuities



This section controls averaging for elemental contouring. It is only available when *Contour Type* is *Elemental*. If *No Average* is selected, contours for each element will be created without consideration to any connected elements. This can lead to a very discontinuous plot but is useful for certain models such as variable thickness plate models to speed the data conversion process. It is also useful to obtain a graphical representation of the pure data, both centroidal and corner data, since only pure data is plotted. If this option is not checked, the user can create averaged elemental contours, and must therefore choose the type of discontinuities across which they do not want to average.

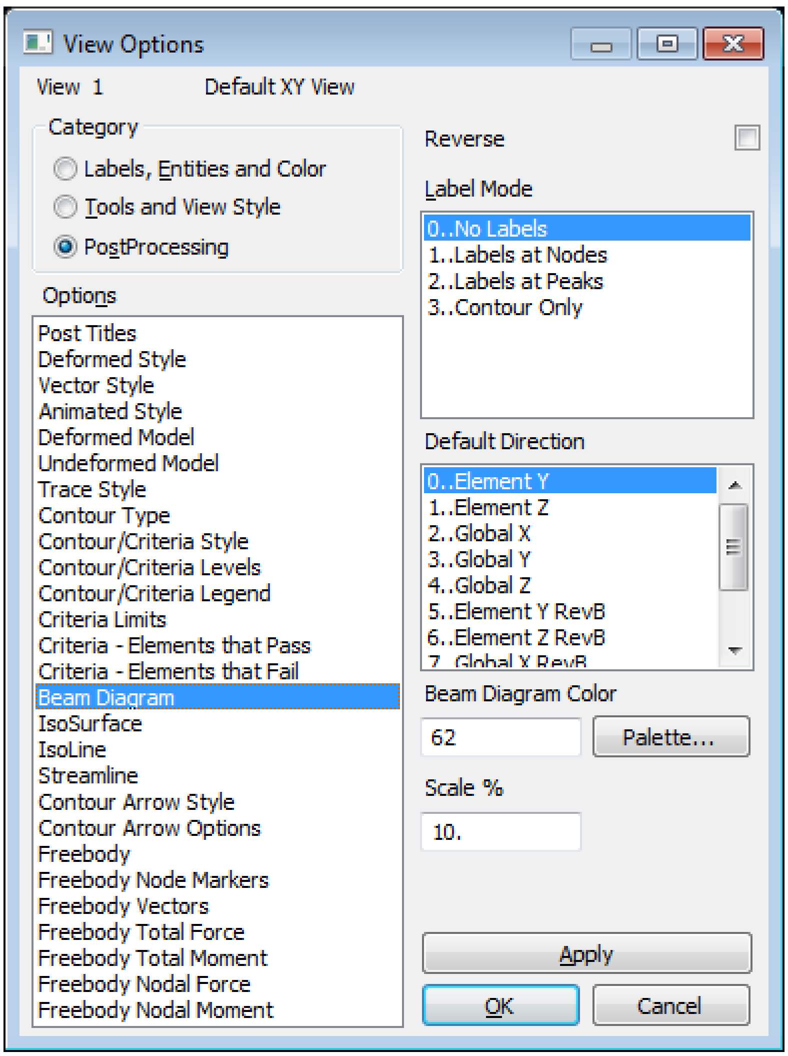
Valid discontinuities include *Property, Material, Layer, Color,* or *Angle*. If *Angle* is selected, you must enter a tolerance. This can be very important with plate models that have intersecting edges. For example, you do not want to average stresses of plates that intersect at right angles.

If *Property* is selected, the material option will be grayed since *Property* is a more discrete choice than *Material* (a material can be on multiple properties, but typically a property can only reference one material). Again, you do not typically want to average across material or property boundaries. If *Property* is off, you can select to use *Materials* as the break.

In addition, layers and colors are also available since many users separate their model into specific key areas based upon layer or color, even if they contain the same property.

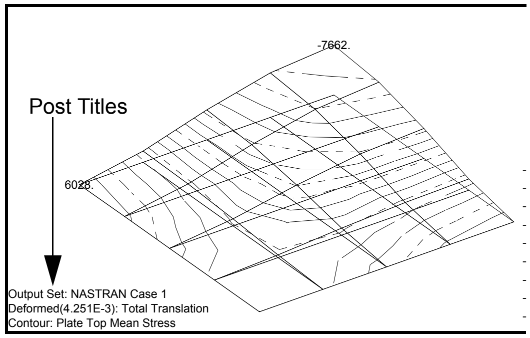
Specifying Detailed Post-Processing Display Options

Options for controlling the detailed aspects of post-processing can be found in the *View Options* command. Each graphics window can have its view options modified independent of other views. The number and depth of the various view options is such that a full discussion of each is not possible in this manual. For more information, see Section 8.3, "View Options - PostProcessing" in *FEMAP Commands*.



Post Titles

Controls whether an additional legend is displayed for deformed or contour views. This legend contains information about the output set and output vectors which are displayed. You can position the legend in any of the eight locations. Make certain it does not overlap the view legend or the contour/criteria legend.



Deformed Style View Options

The following options control the deformed plot:

- *Deformed Style* - determines on-screen scale of deformations.
- *Vector Style* - controls % of vectors displayed and arrowheads for Deformed Vector Plots as well as color for Deformed Vectors.
- *Animated Style* - number of frames, delay, and shape of the animation (Sine, Linear, etc.)
- *Deformed Mode*l - controls colors for *Deformed Style* display.
- *Undeformed Model* - can display or remove the undeformed model, as well as set its color.
- *Trace Style* - controls labeling and display of trace plots.

Contour Style View Options

- *Contour Type* - controls type of contour and Contour Fill mode levels option (for more information, see "Contour Options" above).
- *Contour/Criteria Style* - allows choice of solid/filled or line contours, controls data conversion between nodal and elemental data, and controls labeling options.
- *Contour/Criteria Levels* - controls number and spacing of levels for a contour or criteria plot. You may define your own levels (and colors), or have FEMAP automatically scale the plot.
- *Contour/Criteria Legend* - controls style, color, and visibility of the contour legend.
- *Criteria Limits* - selects criteria for criteria plots.
- *Beam Diagrams* - controls direction of beam diagram plots.
- *Criteria-Elements that Pass* - controls display of elements and their values that pass criteria.
- *Criteria-Elements that Fail* - controls display of elements that fail criteria.
- *IsoSurface* - controls color and mode for isosurface plots.
- *IsoLine* - controls color and mode for isoline plots.
- *Streamline* - controls color and mode for streamline plots.
- *Contour Arrow Style* - allows you to choose where vectors are located, displayed as “wireframe” or “solid” arrows, and if the additional contour vector(s) and *Arrow Type* are automatically selected by the program. You can also specify an overall length, specify a minimum value for display, and if they are shown using “contour colors” or “arrow colors”.
- *Contour Arrow Options* - allows you to choose whether the length of each arrow is adjusted based on magnitude, if the arrows are labeled, how many digits to display, and a minimum length, as a percentage of a *Length* specified in *Contour Arrow Style*.

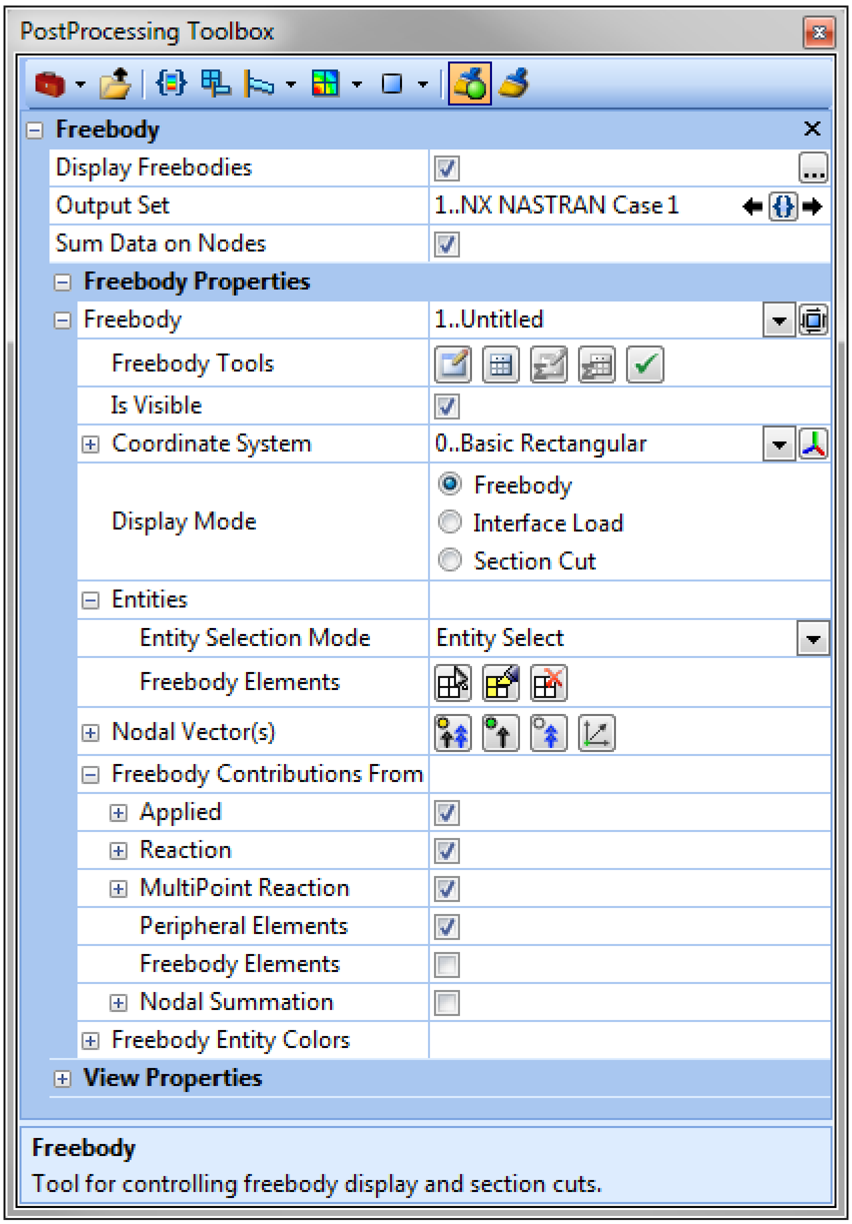
For information about the *Freebody* View Options, see Section 5.11.1.1, "Freebody Plots".

5.11.1.1 Freebody Plots

Freebody information for an entire body, selected elements, or a specific group of elements can be displayed in FEMAP. The Freebody display can be performed at any time, whether you are showing a deformed and contour plot, or a simple undeformed plot. You can setup and control the Freebody display through the *Freebody* tool in the *PostProcessing Toolbox*.

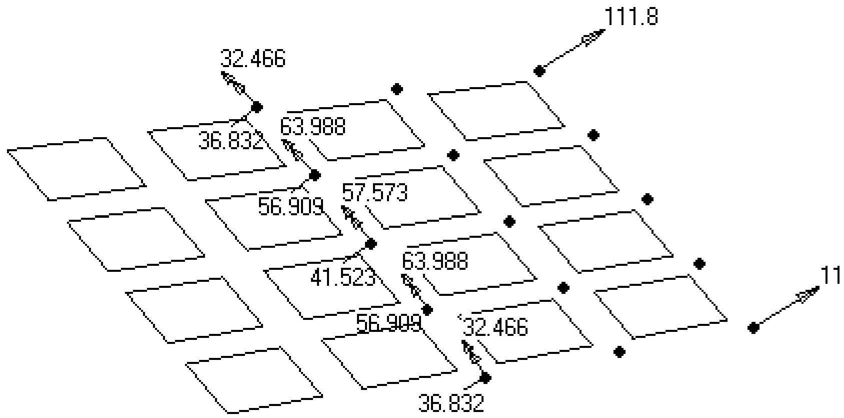
Note: The *Freebody Elements* and *Peripheral Elements* in the *Freebody Contributions From* section will only be available if you have recovered grid point force balance from Nastran. If you are not using Nastran, or have not recovered the grid point force balance, you will only have access to the *Applied* and *Reaction* loads (including *MultiPoint*), thereby limiting the overall usefulness of freebody displays.

The *Freebody* tool in the *PostProcessing Toolbox* is shown here. Following is a general overview of using this command. For more information, see Section 7.2.3.3, "Freebody tool".

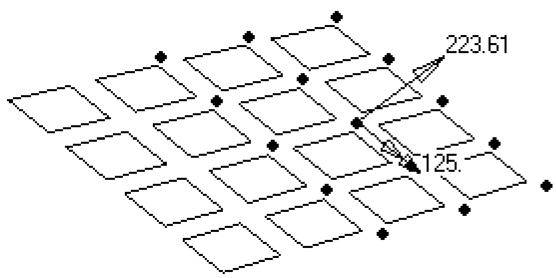


The most typical use of this command is to examine the forces across a specific interface in your model to check the load transfer path, examine the results and their validity, and possibly even create loads to drive another analysis. To do this, elements should be selected on one side of the interface. In the *Freebody* tool, element selection (and optionally node selection for "Interface Loads") is done in the *Entities* section. When the *Entity Selection Mode* to "Entity Select", simply click the *Add Elements* icon button to use the standard selection dialog box. When the *Entity Selection Mode* to “Group Select”, choose an existing group from the drop-down list (“-1..Active” will always use the “Active” group).

The picture shown below is a simple example of a freebody diagram using *Display Mode* set to “Freebody” in the *Freebody* tool. The eight elements in the two columns on the right of the diagram were selected (could also be placed in a group), and the freebody displayed. The result was a diagram containing the external plate forces at the interface, and the original applied loads on the ends (there were no constraints on this section of the model).



By turning all options “on” in the *Freebody Contributions From* section of the *Freebody* tool, you could check that the total loads summed to zero to verify that equilibrium conditions were met and “leaking” of loads did not occur.



If you change the *Display Mode* to “Interface Load”, you then will want to specify a *Location* to calculate the *Total Force* and *Total Moment* of the force balance at a particular point in space.

When *Display Mode* is set to *Section Cut*, a plane is defined using the standard plane definition dialog box, then elements and nodes are automatically selected for calculation of the “Interface Load” based on both proximity to the specified plane and other selected options.

You could also examine results using component vectors and/or in any desired coordinate system. If you wanted to create loads at particular locations to replace portions of your model, you could employ the *Model, Load, From Freebody* command to automatically create these loads.

Freebody View Options

There are several view options to control the overall display:

- *Freebody* - overall on/off for Freebody display and Label options (labels appear at *Location* of “Total” vectors).
- *Freebody Node Markers* - controls *Color Mode* and *Symbol Size* of the node markers.
- *Freebody Vectors* - controls the *Length*, *Label Mode*, and *Label Format* of all freebody vectors and whether their lengths are adjusted.
- *Freebody Total Force/Freebody Total Moment* - specify the *Color Mode*, *Vector Style*, and *Factor*
- *Freebody Nodal Force/Freebody Nodal Moment* - specify the *Color Mode*, *Vector Style*, and *Factor*

5.11.2 XY Plotting using the Charting pane

All XY plotting of output and functions is done using the *Charting* pane. Any number of *Chart* entities may exist in a FEMAP model and any number of *Data Series* entities may be plotted on an individual *Chart* entity. Only one *Chart* at a time may be displayed by the *Charting* pane and selection of which *Chart* to display is controlled via the *Chart Selector* drop-down list.

A *Chart Manage*r may be accessed for creation, editing, copying, deleting, and renumbering of *Charts*. A *Data Series Manager*, with similar functionality, exists for *Data Series* as well.

See Section 7.2.4, "Tools, Charting" in the *FEMAP Commands* manual for detailed information about using the *Charting* pane effectively.

Controlling an XY Plot

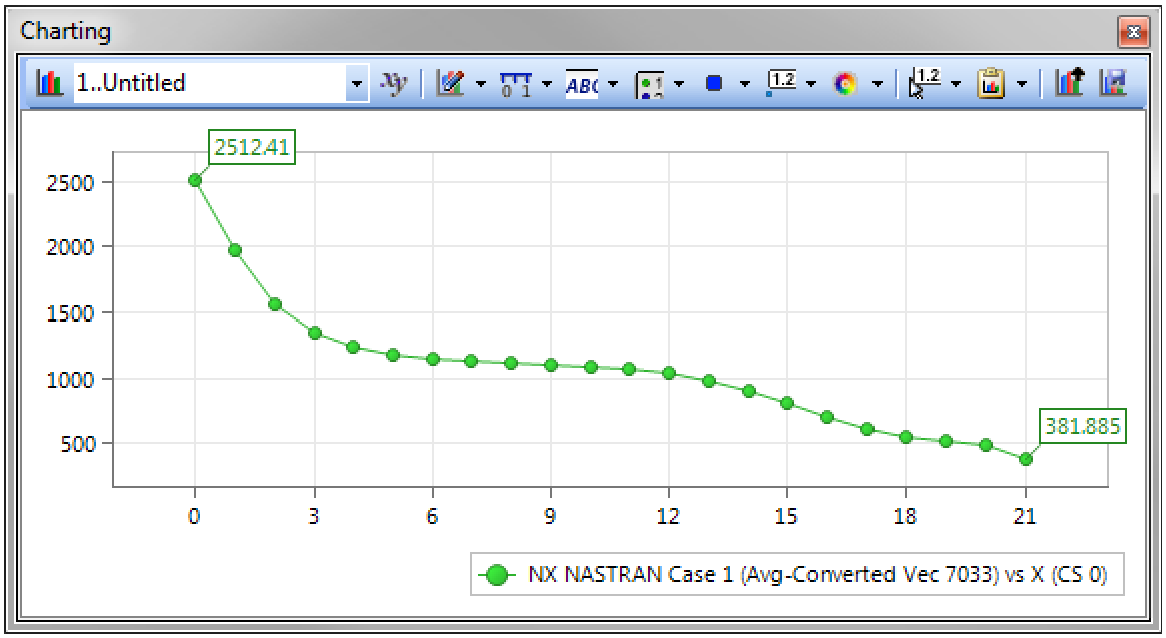
Creation of a new *Chart* and overall control of the current *Chart* is handled by the tabbed *Charting* dialog box. Each tab (*Chart Options*, *Chart Axes*, *Chart Title*, and *Labels and Marker*) controls a different part of the *Chart*.

The *Chart Data Series* dialog box is used to specify the *Data* (*Type*, *Group*, *Output Data*, *Location*, *Position*, or *Function*) and *Style* (*Labels*, *Markers*, and *Colors*) for each *Data Series*.

In addition, the *Chart Options* icon menu allows creation of a new *Data Series*, selection of which *Data Series* to display in the current *Chart*, and selection of a particular *Style*. A number of other icon menus across the top of the *Charting* pane control the overall display and options for *Chart Axes*, *Chart Title*, *Chart Legend*, *Data Series Markers*, *Data Series Labels*, and *Chart Colors*. Icons for *Show Tooltips*, *Copy to Clipboard*, and *Load From Library/ Save to Library* complete the *Charting* pane toolbar.

Finally, a context-sensitive menu for the current *Chart* is available, along with context-sensitive menus which appear when the cursor is placed over a particular entity (*Data Series*, *Axes*, *Legend*, *Chart Title*, *Markers*, or *Labels*) in a *Chart*.

Example plot of Data Series:

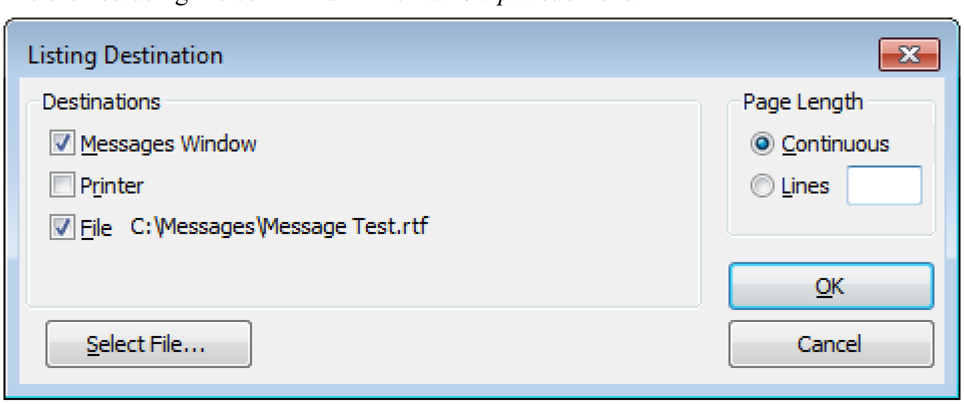


5.11.3 Reporting Results

In addition to the graphical post-processing capabilities of FEMAP, there is also a powerful set of report based tools for examination of FEA results.

Directing Output

Reports are created using the command in the *List Output* submenu



By default, all listings go to the *Messages* window. You can also direct listings to a printer and/or a file. To control where listings appear, choose *List, Destination* and select the desired options.

Note: Make certain to toggle off listings to printer or file when you finish listing the desired information. FEMAP will continue to send all listings to whatever destinations have been chosen until they are turned off.

Listing Formatted Output

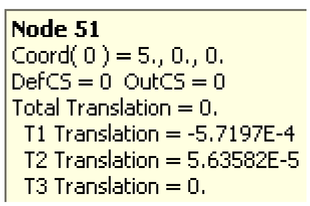
The most powerful commands associated with listing output are *List, Output, Standard* and *List, Output, Use Format*. Both are used to process the nodal and elemental data recovered from a finite element analysis and repackage that data into standard formats or ones you define, and then list that data in printed format.

One of the easiest methods of creating your own format is to load a standard format and then edit it. For more details, see Section 7.5, "List Menu Commands" in *FEMAP Commands.*

Querying Your Model

There are two methods to quickly query your model for post-processing information: the *List, Output, Query* command, and the *Show Tooltips* command. If you would like to examine large amounts of data for a single entity, simply use the *List, Output, Query* command. This command provides a quick method for retrieving the output results for a particular node or element, or group of nodes or elements in your model. The results, as always, will be written to the List File Destination area(s).

5.11.3.1 Show Tooltips



To access the *Show Tooltips* capability, you must select the command from either the *Quick Access Menu*, or the *Select Toolbar's* Selector Modes Menu. When toggled on, this mode will be available whenever the *Select Toolbar* has an active entity or you are using a dialog box to select a certain type of entity. If you are graphically post-processing while *Show Tooltips* is activated and have an active selection entity in the *Select Toolbar*, the exact information that is being used to create the graphic is also displayed in the pop-up windows. The same is true when selecting entities with a dialog box while graphically post-processing.

Note: You can set how long your tooltips will take to appear and how long they will remain displayed on your screen using *File, Preferences*, clicking the *User Interface* Tab, then assigning values for "Tooltip Delay" and "Tooltip Duration". Both values should be entered in tenths of a second.

The following commands only work when the *Select Toolbar* is being used with *Show Tooltips* on:

While a "Tooltip" pop-up window is displaying information, if you click the left mouse button, the information will be sent to the Entity Editor and/or Data Table dockable panes, as long as the panes are visible in the FEMAP interface AND unlocked.

If you click the right mouse button inside the current "Tooltip", a short menu will appear:

List - sends the information in the Tooltip to the *Messages* window. Using this capability, you can quickly walk around the model and recover important information at specific nodes and elements. You can now copy this information from the *Messages* window or use *List, Destination*, to send the data to Rich Text Format file outside of FEMAP. Either method can help you can easily create a report in another program.

Convert To Text - creates a text entity identical to the *Show Tooltips* box at that location to help annotate your model. You MUST have *Text* visible to see the yellow text entities. Text can be made visible using either *View, Options* or *View, Visibility.*

Hint: Pressing Alt + clicking the right mouse button in the graphics window will bring up the *Quick Access Menu* instead of the context sensitive menu when there is an active entity in the *Select Toolbar*. Using this method, you can toggle the *Show Tooltips* command on and off without having to use the *Select Toolbar's* Selector Modes Menu.

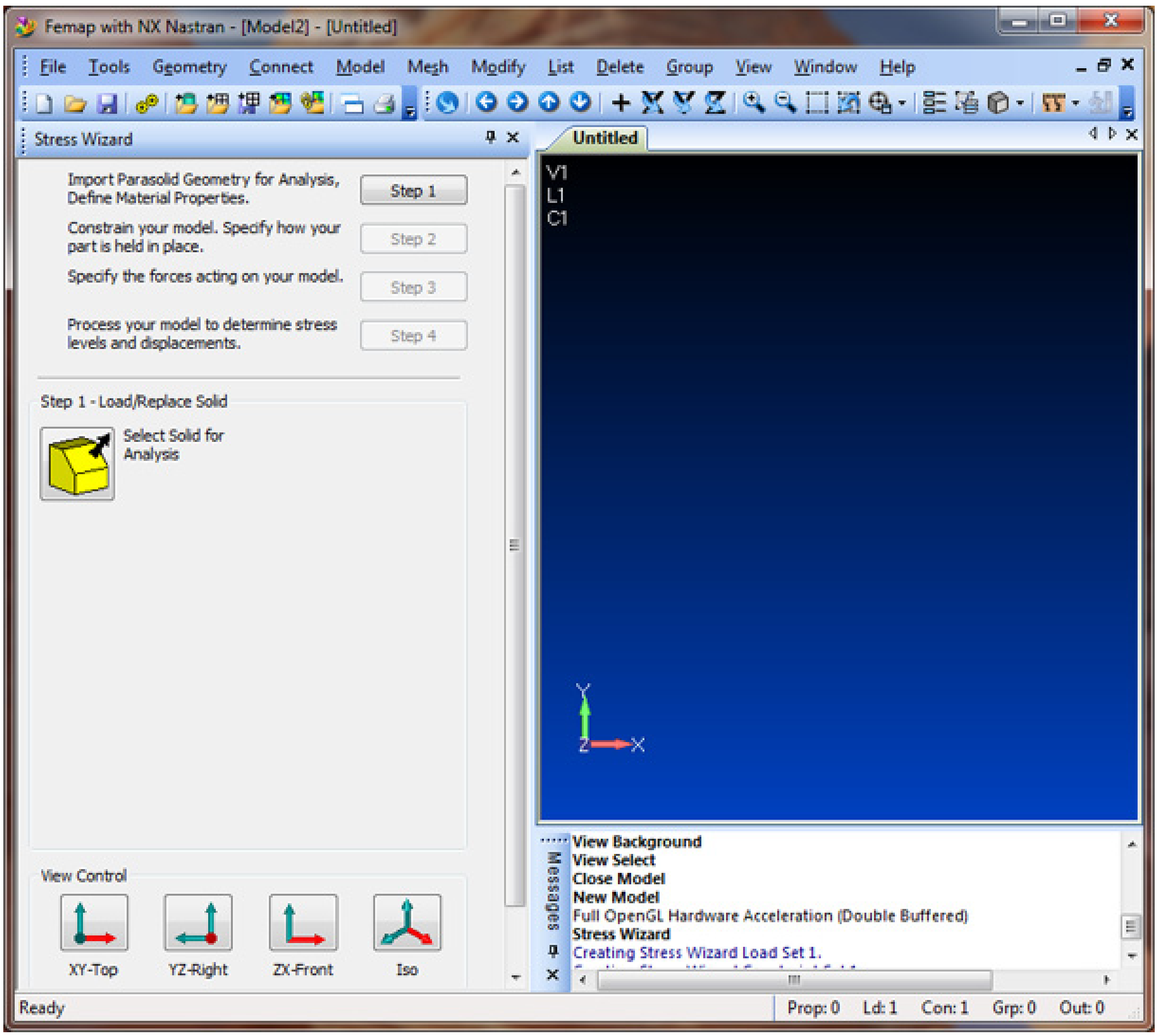
For more information on see Section 7.3.1.6, "Tools, Toolbars, Select" of the commands manual and Section 4.2.3, "Quick Access Menu (Right Mouse Button)"

5.12 Stress Wizard

The Stress Wizard (SW) provides you with quick insight into the mechanical behavior of engineering parts. Through a simple four-step process, the SW makes it possible to connect to a single solid or multi-solid assembly, specify how the solid/assembly is held, how it is loaded, and recover the resulting deformed shape and stress distribution. In reality, the SW provides access to several different areas of FEMAP functionality from within a dockable pane. The Stress Wizard does not add any functionality over what is offered within other FEMAP commands; it simply consolidates the commands required for the pre- and post-processing and analysis of single solid parts or multi-solid assemblies. For more information on specific commands in the Stress Wizard see Section 7.4.6, "Tools, Stress Wizard".

A Simple Analysis Example using the Stress Wizard

Before we get into the details, a step-by-step walk-through example will be used to familiarize you with the Stress Wizard. Start the Stress Wizard (*Tools, Stress Wizard* from the FEMAP menu) the screen should look like:

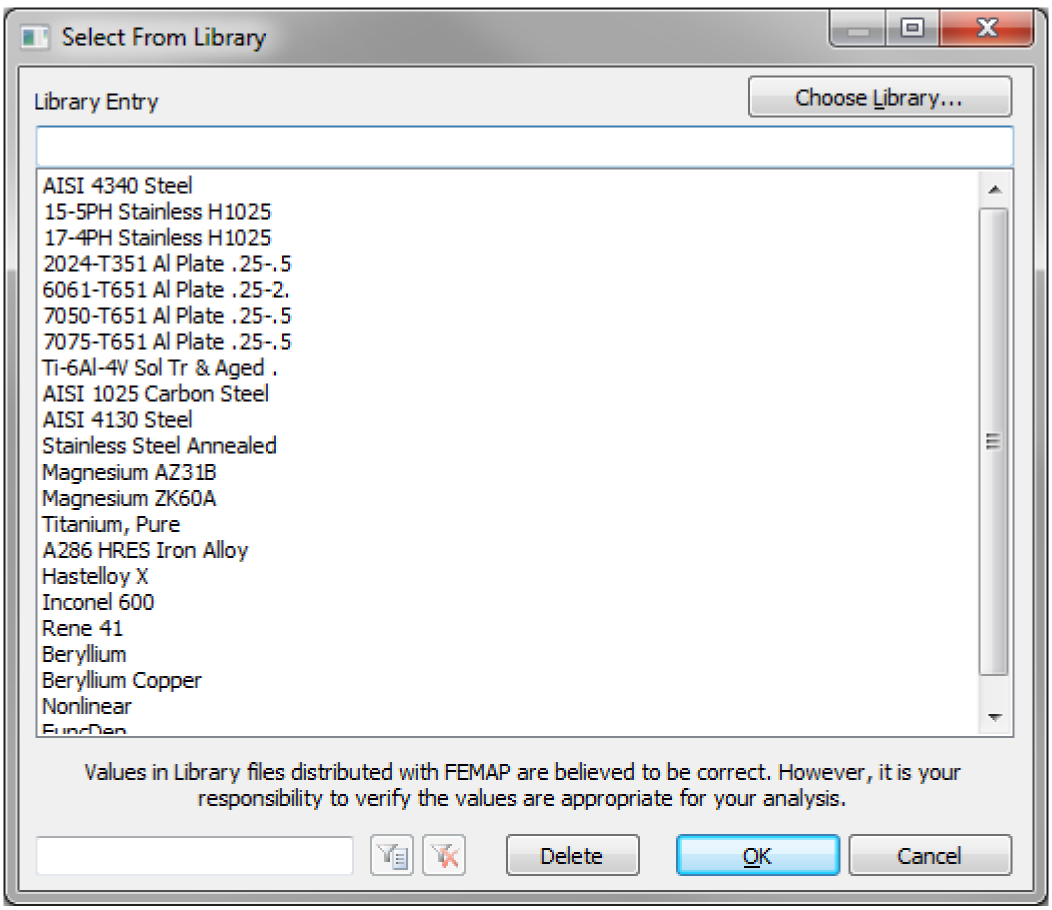


5.12.1 A Simple Analysis: Step 1 - Importing the Geometry

Step 1 of the Stress Wizard has only one option, to import geometry. Select the button shown below to select a Parasolid geometry file for import.

Navigate to the directory where the FEMAP example files have been installed and select the file "BATH_125.X_T" for import. Once selected, press the "Open" button to load the part.

You will now be prompted to select a material for this part, simply select one from the library.



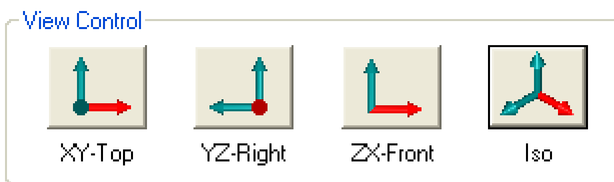
NOTE: Do not worry if the material that you wish to use for your part is not in the library, you will have the option of editing the material properties later on, and the ability to add the new edited material to the library for future use.

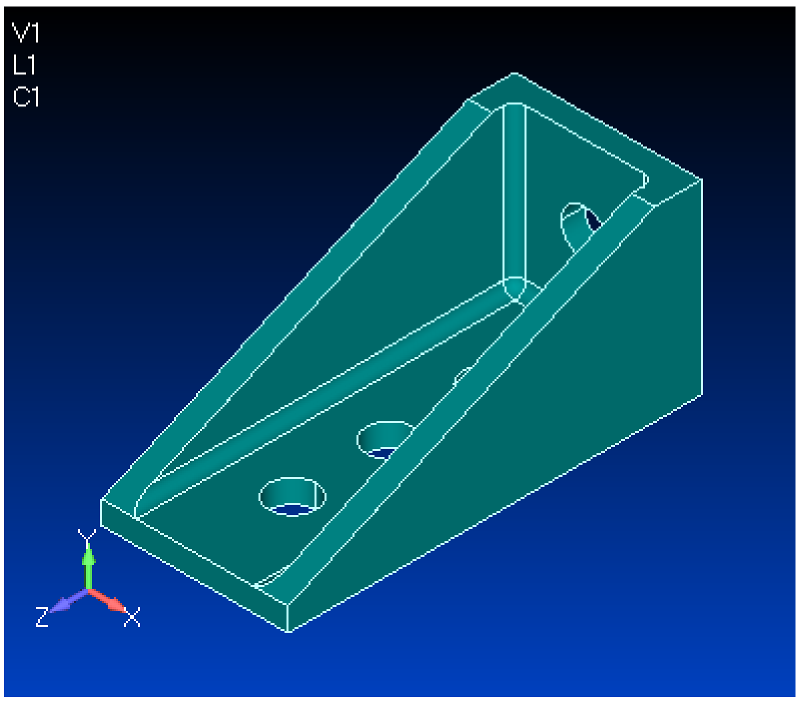
After you press *OK* in the Material *Select From Library* box, you will see the Stress Wizard performing some of the traditional Finite Element Analysis tasks in the background. You will see messages regarding the meshing of the part.

5.12.2 A Simple Analysis: Step 2 - Constraining the Model

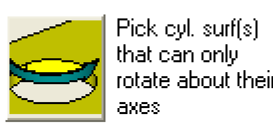
After the part has been imported in Step 1, the Stress Wizard automatically jumps to Step 2. In Step 2 we will specify how the part is constrained.

To get the part into an isometric view, you can press and drag the left mouse button in the graphics window and rotate the model as shown below, or press the “Iso” button in the “View Control” section of the Stress Wizard dialog box.





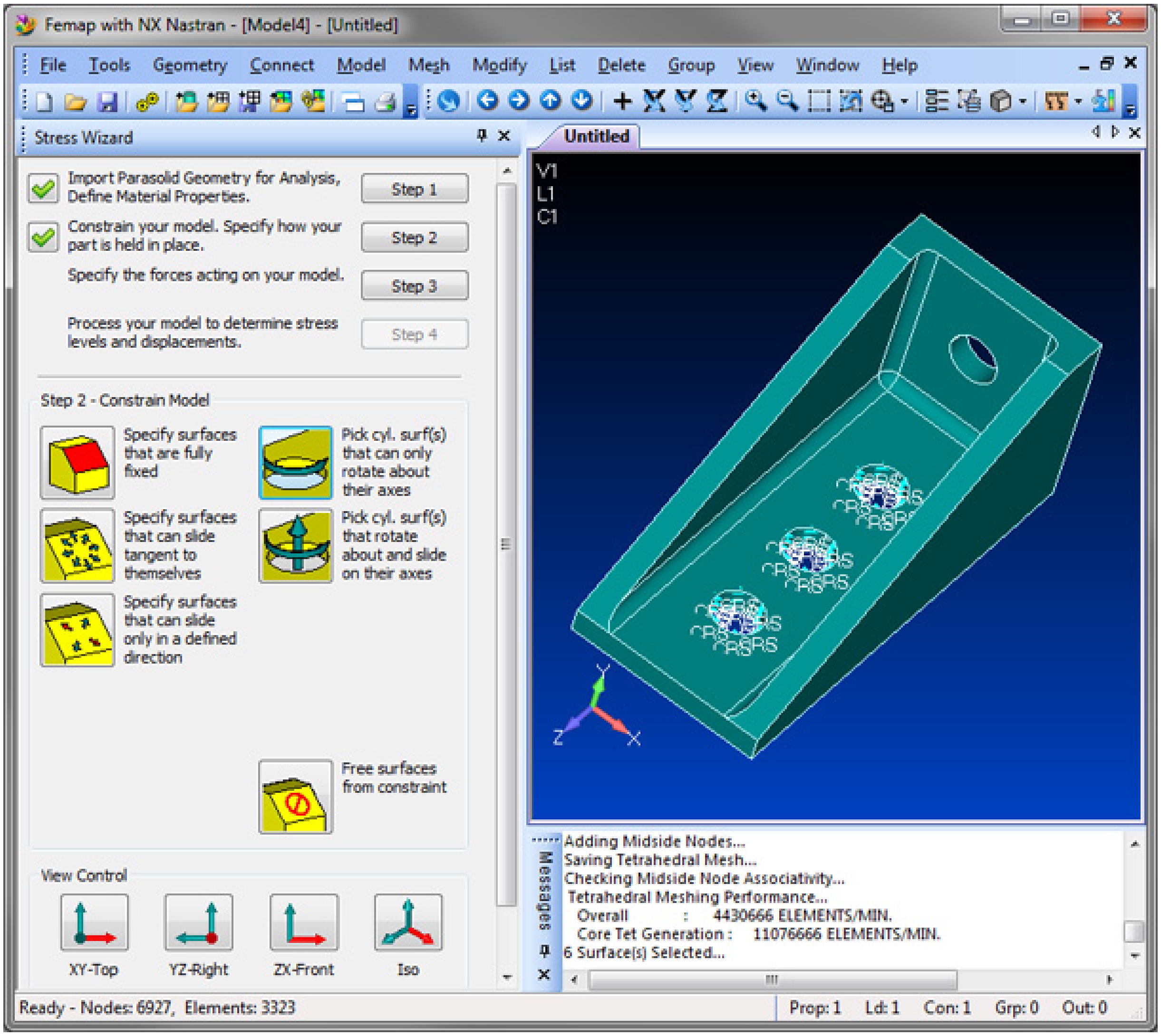
Step 2 provides access to constraining the surfaces of your solid part. Once the finite element analysis is started, FEMAP will convert surface based constraints into nodal based constraints. For this example, we will try to approximate a bolted connection in the three bottom holes of our part. Press the button seen below “Pick cyl. Surf(s) that can only rotate about their axes”. This will create nodal constraints on cylindrical surfaces where the



nodes will not be allowed to move radially from the center axis of the cylinder and not allowed to move in the plus or minus axial direction.

NOTE: A hole constrained this way is a significant engineering approximation. Accurately modeling the constraint of a pre-loaded bolted connection is beyond the capability of this simple wizard. Please be advised that the stress distribution around this constrained area will not be correct.

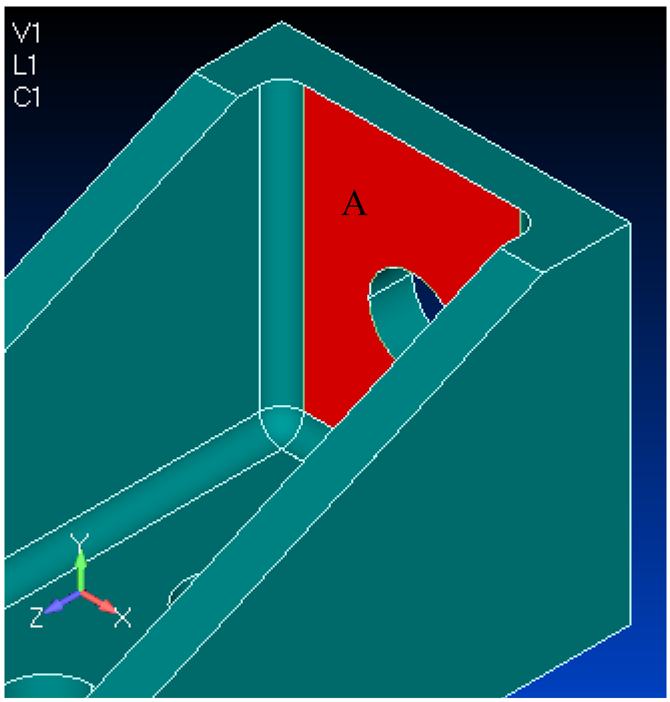
After you have selected these six surfaces and pressed OK, each will receive a small “CRS”, the CRS indicates that the “C”ylindrical surface has been constrained “R”adially and from “S”liding.



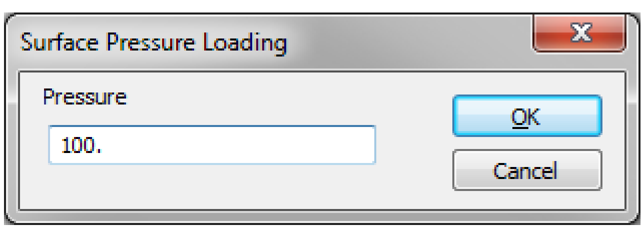
5.12.3 A Simple Analysis: Step 3 - Loading the Model

Press the “Step 3" button to move on to loading this solid model. We will apply a pressure load to the inside face at the back of the fitting. Press the button shown below to apply a pressure load to one or more surfaces.

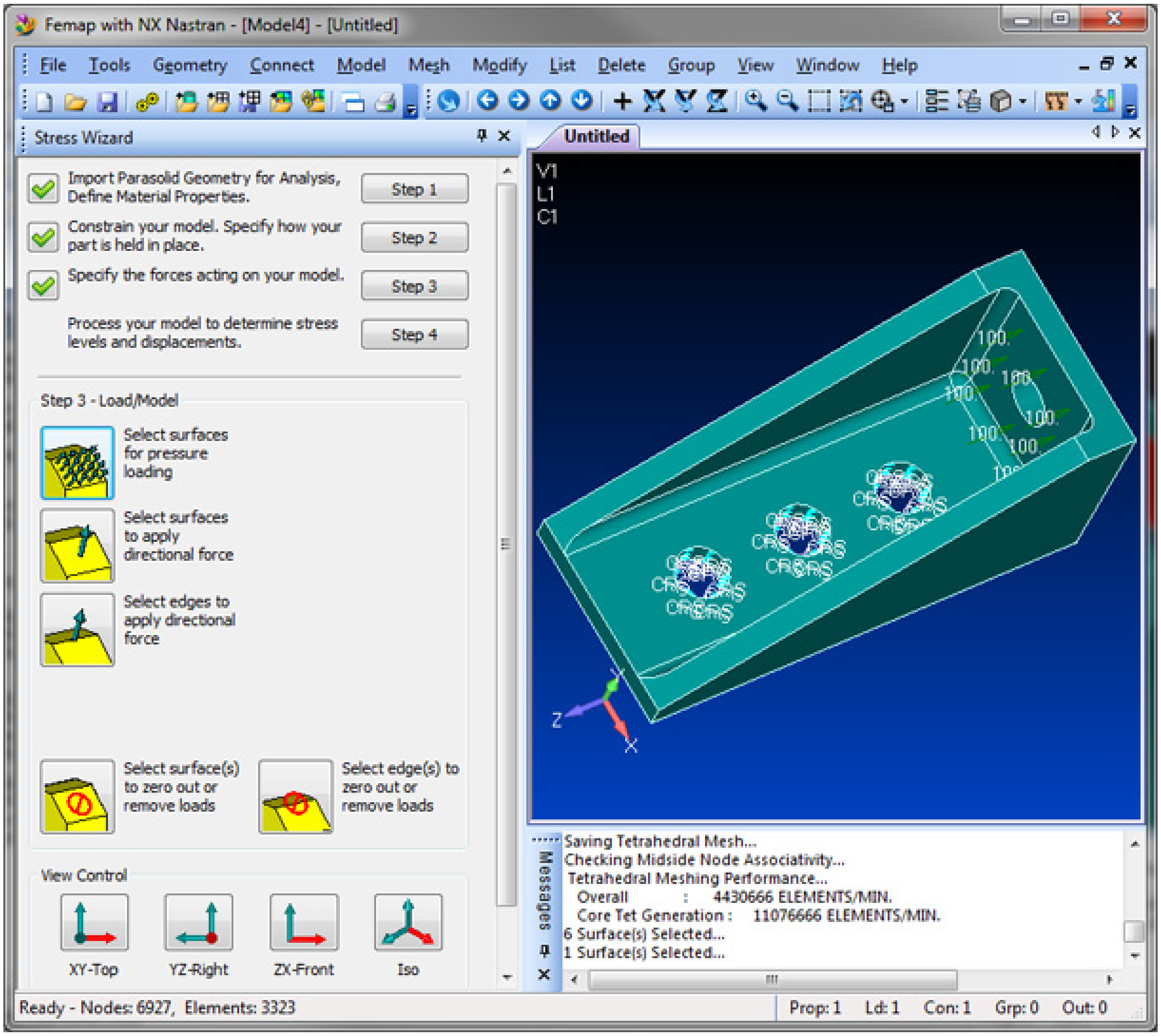
FEMAP will now prompt you with the standard entity selection box and ask you to pick a surface. Pick the inside face at the back side of our fitting (surface **A** shown in red below).



Press “OK” in the standard entity selection box, you will then be prompted for a pressure value, enter 100 and press “OK” to continue.



You will now see the pressure load on the surface selected. Remember, if you make a mistake constraining or loading your model, you can use the Tools - Undo (Ctrl-Z) feature of FEMAP to undo your last couple of actions.



5.12.4 A Simple Analysis: Step 4 - Analyzing and Post-Processing

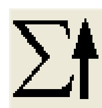
At this point we are ready to run this model and recover deflections and their associated stresses. The Stress Wizard does provide feedback that the first three steps of the FEA process are complete with the red check marks next to Steps 1, 2 and 3. Press Step 4 so that we can run this model.

Before the model has been run (an input file is created and submitted to any of the FEMAP supported batch solvers), there is only one option available in Step 4 “Run this Model and Recover Answers”. In order for the automatic execution of a solver to be possible, you must first have configured FEMAP in File-Preferences to point to specific FEA solver. Press the run button to process your FEA model.

If your run was successful, the remaining options in Step 4 will be available. Including displaying contours of the Stress or Displacement distributions.

One that I will point out that is extremely useful, but not in the usual colorful stress/deformation plots, is the ability to sum reaction forces on the surfaces of your solid model. With this tool, you can recover the forces associated with holding your model down, this is extremely useful in determining if the fasteners or welds used to hold your part are sufficiently strong.

To try this option, press the “List Reaction Forces on Surface(s) button. Next, select the two halves of one of the base holes that you constrained in Step 2. FEMAP will display some dialog boxes detailing the Global X, Y and Z values of force that were required to hold the selected surfaces as specified.



Update the Solid Model and Re-run

With four simple steps, the Stress Wizard has allowed us to go from solid model to answers. The most powerful feature is the ability to update the solid model in your CAD system, and rerun the analysis to determine the effect of any design changes. To experiment with this capability, Press the Step 1 button and return to Step 1. You will now notice that the caption for the button reads “Select Updated Solid for Analysis”. Through FEMAP's ability to import an updated solid and slip it under the boundary conditions and loads previously specified, we can bring in a design update and quickly evaluate its consequences. Press the “Select Updated Solid for Analysis” button and import BATH_25.X_T. This file is identical to our first one with an increased blend radius on the inside of our fitting.

You will see the solid change on screen. The constraints and loads previously specified are transferred to the updated solid. You can now go directly back to Step 4, re-run the model, and look at the results of the design change.

Element Reference

This topic describes the FEMAP element library, the geometry used to create the elements and apply loads, and the properties which can be specified. The descriptions given for the various element types define typical characteristics of the elements as they are translated to various analysis programs. Check your analysis program documentation for additional capabilities or limitations of each element type in that program.

There are several element types based on the general shape of the elements as well as other types and “connector elements. They are divided into 4 sections:

- Section 6.1, "Line Elements"
- Section 6.2, "Plane Elements"
- Section 6.3, "Volume Elements"
- Section 6.4, "Other Elements"

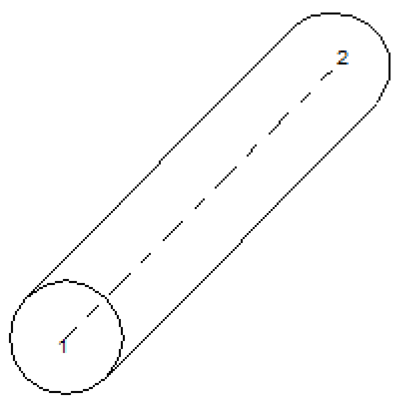
6.1 Line Elements

All of the elements in this section structurally connect two nodes. The different types represent different structural conditions.

6.1.1 Rod Element

Description

Uniaxial element with tension, compression and torsional stiffness. It does not have any bending or shear capability.



Application

Typically used to model truss, or other “pin-ended” members.

Shape

Line, connecting two nodes.

Element Coordinate System

The element X axis goes from the first node to the second.

Properties

Area (of cross-section), Torsional Constant, Coefficient for Torsional Stress, Nonstructural Mass/Length.

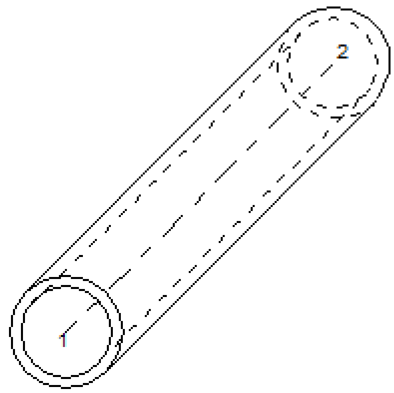
Formulation

None or Hybrid. Hybrid formulation only affects ABAQUS export (hybrid TRUSS).

6.1.2 Tube Element

Description

Variation of the rod element with a tubular cross section. It is also a uniaxial element with tension, compression, and torsional stiffness. Some analysis programs also include bending and shear stiffness when they use this type to represent a pipe.



Application

Often used to model pipes. Also used as a more convenient way to specify properties for a rod element if the cross section is tubular.

Shape

Line, connecting two nodes.

Element Coordinate System

The element X axis goes from the first node to the second.

Properties

Inner Diameter, Outer Diameter, Nonstructural Mass/Length.

Formulation

None or Hybrid. Hybrid formulation affects ABAQUS export (hybrid PIPE) and MARC export (element 14 for hybrid, otherwise 31).

6.1.3 Curved Tube Element

Description

Another tube type element. This element is curved. The neutral axis is an arc, not a line, which goes between the nodes. You can often use multiple tube elements, arranged in an arc, instead of this element.

Application

Modeling of bends and elbows in piping systems, or other curved members.

Shape

Arc, connecting two nodes.

Element Coordinate System

Same as beam/curved beam element. The element is curved in the elemental XY plane, with the outward radius pointing toward the third node (or in the direction of the orientation vector). For a picture of the element definition, see Section 6.1.7, "Curved Beam Element".

Properties

Inner Diameter, Outer Diameter, Nonstructural Mass/Length.

Formulation

None.

Additional Notes

Unlike the beam element, offsets, stress recovery locations, and releases are not supported for this element type.

6.1.4 Bar Element

Description

Uniaxial element with tension, compression, torsion, and bending capabilities. The more general beam element is often used instead of this element. The figure at the end of this section, defines both element types. For some analysis programs, FEMAP translates both types to the same element type.

Application

Used to model general beam/frame structures.

Shape

Line, connecting two nodes. A third node can be specified to orient the element Y axis.

Element Coordinate System

The element X axis goes from the first node to the second. The element Y axis is perpendicular to the element X axis. It points from the first node toward the orientation (or third) node. If you use an orientation vector, the Y axis points from the first node in the direction of the orientation vector. The element Z axis is determined from the cross product of the element X and Y axes.

Properties

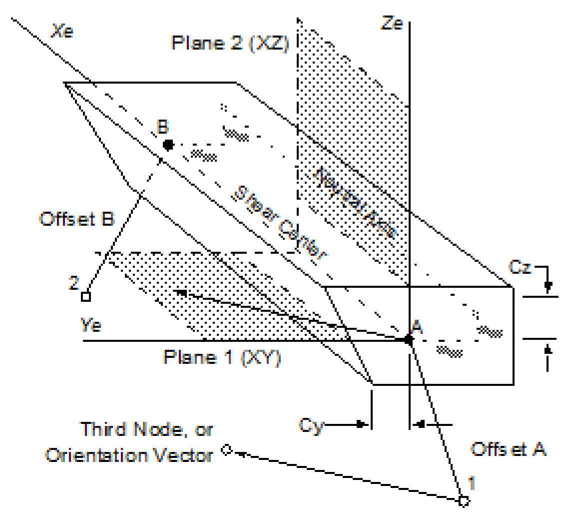
Area, Moments of Inertia (I1, I2, I12), Torsional Constant, Shear Areas (Y, Z), Nonstructural Mass/Length, Stress Recovery Locations. All required input properties for this element can be automatically calculated for standard or arbitrary shapes by using the FEMAP beam property section generator (accessed under *Model, Property, Shape*). The Shear Areas calculated by the beam property section generator and the input to FEMAP are the effective areas for shearing, not a shear factor. If you are inputting values directly and have a shear factor, simply multiple it by the actual area to obtain the shear area.

Formulation

Nine available formulations for DYNA (1..Hughes-Liu is default) defining value for ELFORM on *SECTION_BEAM card . Standard (MARC - 98, ABAQUS - B21, B31) or Euler-Bernoulli (MARC - 52, ABAQUS B23, B33) options. The Hybrid formulation option only affects ABAQUS export by adding an “H” to the element name, thereby calling the ABAQUS hybrid form of the element.

Additional Notes

For further descriptions regarding Releases, Offsets and Stress Recovery Locations, see Section 6.1.5, "Beam Element".



Bar / Beam Elements

6.1.5 Beam Element

Description

Uniaxial element with tension, compression, torsion, and bending capabilities. This element can be tapered. You can specify different properties at each end of the beam.

Application

Used to model beam/frame structures.

Shape

Line, connecting 2 or 3 nodes. An orientaion node can be specified to orient the element Y axis.

Element Coordinate System

The element X axis goes from the first node to the second. The element Y axis is perpendicular to the element X axis. It points from the first node toward the orientation (or third) node. If you use an orientation vector, the Y axis points from the first node in the direction of the orientation vector. The element Z axis is determined from the cross product of the element X and Y axes.

Properties

Area, Moments of Inertia (I1, I2, I12), Torsional Constant, Shear Areas (Y, Z), Nonstructural Mass/Length, Warping Constant, Stress Recovery Locations, Neutral Axis Offsets (Nay, Naz, Nby and Nbz). All required input properties for this element can be automatically calculated for standard or arbitrary shapes by using the FEMAP beam cross section generator (accessed under *Model, Property, Shape*). The Shear Areas calculated by the beam property section generator and the input to FEMAP are the effective areas for shearing, not a shear factor. If you are inputting values directly, and have a shear factor, simply multiple it by the actual area to obtain the shear area. If the beam is tapered, you can specify different properties at each end of the element.

Formulation

Nine available formulations for DYNA (1..Hughes-Liu is default) defining value for ELFORM on *SECTION_BEAM card.

Standard (MARC - 98, ABAQUS - B21, B31) or Euler-Bernoulli (MARC - 52, ABAQUS B23, B33) options.

The *Hybrid* formulation option only affects ABAQUS export by adding an “H” to the element name, thereby calling the ABAQUS hybrid form of the element.

Three formulations for ANSYS:

BEAM44 - default. Always sets KEYOPT(7) and KEYOPT(8) to appropriate values for beam releases.

BEAM188/section shape - creates BEAM188 and uses SECTYPE, (section shape) when possible, to have ANSYS calculate cross-section property values. Uses SECTYPE, ASEC if “section shapes” does not exist in ANSYS. Also, always sets KEYOPT(3) = 3

BEAM188/ASEC - creates BEAM188 and SECTYPE, ASEC for all beams with this formulation, then exports cross-section property values calculated in FEMAP. Also, always sets KEYOPT(3) = 3

Additional Notes

You can specify releases which remove the connection between selected element degrees of freedom and the nodes.

Offset vectors defined on the element move the neutral axis and shear center from the nodes. *Neutral Axis Offsets (Y,Z)* defined on the property card move the neutral axis away from the shear center. If there are no neutral axis offsets, the neutral axis and shear center are coincident. If there are no offsets, both the neutral axis and shear center lie directly between the nodes.

Stress Recovery Locations define positions in the elemental YZ plane (element cross-section) where you want the analysis program to calculate stresses.

Specifying moments of inertia for beam (and bar) elements can sometimes be confusing. In FEMAP, I1 is the moment of inertia about the elemental Z axis. It resists bending in the outer Y fibers of the beam. It is the moment of inertia in plane 1. Similarly, I2 is the moment of inertia about the elemental Y axis. If you are familiar with one of the analysis program conventions, the following table may help you convert to FEMAP's convention.

FEMAP	I1	I2
NASTRAN	Izz	Iyy
ANSYS	IZ1	IY1
ABAQUS	I22	I11
MARC	Iyy	Ixx
LS-DYNA	Itt	Iss

6.1.6 Link Element

Description

This element is a rigid link with six spring (bushing) stiffnesses at each end. Link elements are only supported by MSC/pal and CDA/Sprint I.

Application

Can be used to represent members that are very stiff compared to the stiffness of their connections. Can also simulate a rigid connection if you specify large spring stiffnesses.

Shape

Line, connecting two nodes.

Element Coordinate System

The elemental X axis goes from the first node to the second.

Properties

Six stiffnesses at each node.

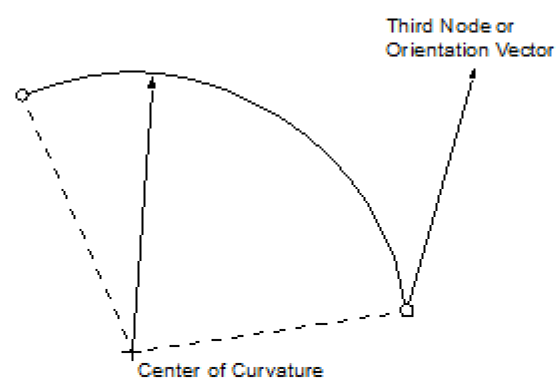
Formulation

None.

6.1.7 Curved Beam Element

Description

Another beam type element. This element is curved. The neutral axis is an arc, not a line, which goes between the nodes. You can often use multiple beam elements, arranged in an arc, instead of this element.



Application

Modeling of bends and elbows in piping systems, or other curved members.

Shape

Arc, connecting two nodes.

Element Coordinate System

Same as beam element. The element is curved in the elemental XY plane, with the outward radius pointing toward the third node (or in the direction of the orientation vector).

Properties

Bend Radius, Area, Moments of Inertia (I1, I2, I12), Torsional Constant, Shear Areas (Y, Z), Nonstructural Mass/Length, Stress Recovery Locations. All required input properties for this element can be automatically calculated for standard or arbitrary shapes by using the FEMAP beam cross section generator (accessed under *Model, Property, Shape*). The shear areas calculated by the beam property section generator and the input to FEMAP are the effective areas for shearing, not a shear factor. If you are inputting values directly and have a shear factor, simply multiple it by the actual area to obtain the shear area.

Additional Notes

For descriptions regarding Offsets and Stress Recovery Locations, see Section 6.1.5, "Beam Element". Note that releases are not supported for this element type.

Formulation

None.

6.1.8 Spring/Damper Element

Description

A combined stiffness (spring) and damper element. It can be either axial or torsional. The DOF spring is an alternative element. The Type specified on the Property for each element is used to determine which element is created.

Application

Used to represent any purely axial, or purely torsional, structural member. Other use is to specify a bushing.

Shape

Line, connecting two nodes. CBUSH Type has circular symbol, “Other” Type has rectangular symbol. Only symbol will appear if nodes are coincident.

Element Coordinate System

The element X axis goes from the first node toward the second.

Type on Property set to CBUSH:

If *Orientation* is set to *CSys* on the element itself, then the element Csys is equal to the selected Csys. If instead, *Orientation* is set to *From Property* on the element AND *Orientation CSys* on the property is enabled, then the element Csys is equal to the selected Csys on the property. If the Orientation Csys is not used, then the element Csys is defined with the X axis going from the first node to the second. The element Y axis is perpendicular to the element X axis. It points from the first node toward the orientation (or third) node. If you use an orientation vector, the Y axis points from the first node in the direction of the orientation vector. The element Z axis is determined from the cross product of the element X and Y axes. The BUSH element also has offsets that are defined in the output coordinate system.

Properties

If Type of referenced Property is set to Other (NASTRAN CVISC/CROD), enter a Stiffness and/or Damping.

If Type is CBUSH, then Stiffness, Damping, and Structural Damping values can be defined for individual degrees of freedom, along with Spring/Damper Location (can also be specified on element), Orientation Csys (can also be specified on element), Stress/Strain recovery coefficients. For Frequency or nonlinear analysis function dependence can be defined for stiffness and damping values.

Formulation

No longer used. They have been replaced by the Type specified in the Property referenced by each element. In Previous versions of FEMAP, 2 formulations were available for NX Nastran and MSC.Nastran. “0..Default” would write a CROD if stiffness was defined on the property or a CVISC if a damping value was defined. “1..CBUSH” would write the spring element as a CBUSH and the corresponding property as PBUSH and PBUSHT.

6.1.9 DOF Spring Element

Description

A combined stiffness (spring) and damper element. This element connects any (of six) nodal degree of freedom at the first node, to any nodal degree of freedom at the second node. The spring/damper element is an alternative element for axial members.

Application

Used to connect two degrees of freedom with a specified stiffness. Depending on the degrees of freedom and the position of the nodes, this can be an axial member or something much more complex.

Shape

Connects two nodes. Drawn as a line with a "jagged" symbol by FEMAP. Only symbol will appear if nodes are coincident.

Element Coordinate System

Determined by nodal degrees of freedom.

Properties

Degree of Freedom (at each node), stiffness, damping.

Formulation

There are 2 formulations available for NX Nastran and MSC.Nastran. "0..Default (CELAS2/CDAMP2)" will write a CELAS2 or CDAMP2 which have both "property" (i.e. stiffness value in the "K" field for CELAS2 or damping value in the "B" field for CDAMP2) and "connection" (i.e., node/grid IDs) information in a single Nastran entry. When "1..CELAS1/CDAMP1" is chosen, a CELAS1 or CDAMP1 will reference an appropriate Property (PID) for a spring (PELAS) or damper (PDMAP). The PELAS and PDAMP are not written at all when using the default formulation.

6.1.10 Gap Element

Description

A nonlinear element which has different tension, compression, and shear stiffnesses. Check your analysis program for further descriptions. The exact capabilities of this element type vary widely between analysis programs.

Application

Used to represent surfaces or points which can separate, close, or slide, relative to each other.

Shape

Line, connecting two nodes.

Element Coordinate System

The element X axis goes from the first node to the second. The element Y axis is perpendicular to the element X axis. It points from the first node toward the orientation (or third) node. If you use an orientation vector, the Y axis points from the first node in the direction of the orientation vector. The element Z axis is determined from the cross product of the element X and Y axes.

Properties

Initial Gap, Compression Stiffness, Tension Stiffness, Transverse Stiffness, Y and Z Friction Coefficients, Preload Force, Interface Plane Normal (ABAQUS Only), Interface Width/Area (ABAQUS Only). Many of these properties are not supported by all analysis programs.

Formulation

None.

6.1.11 Plot Only Element (Line)

Description

This element is nonstructural. It does not add any stiffness to your model. It is only used for plotting.

Application

Used to represent structural features that are not being analyzed, but that aid in the visualization of the model. Plot-only elements are also used by the ABAQUS interface to create interface elements and rigid bodies.

Shape

Line, connecting two nodes.

Element Coordinate System

None

Properties

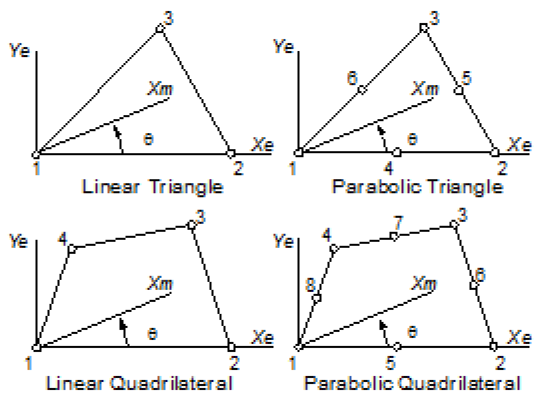
None

Formulation

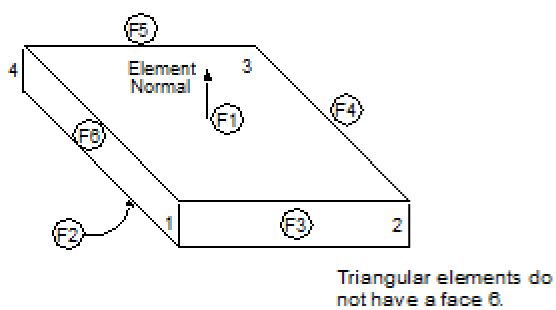
None.

6.2 Plane Elements

The plane elements are used to represent membrane, shell, and plate structures. They all follow the same shape and numbering conventions. The simplest formulation of these elements are just a three-noded triangle and a four-noded quadrilateral. In addition, six-noded "parabolic" triangles and eight-noded "parabolic" quadrilaterals are also available.



In most cases, loads on plane elements will be applied to face 1. In this case positive pressure acts in the same direction as the face normal (as determined by the right-hand rule). Conversely, if loads are applied to face 2, their positive direction will be opposite to the face normal. Therefore a positive pressure on face 2 is equivalent to a negative pressure on face 1. If you need to apply edge loads, they can be applied to faces 3 through 6 as shown. Their positive direction is inward, toward the element center.



Whenever possible, you should try to use elements which closely resemble equilateral triangles or squares. These shapes will usually result in the best analysis accuracy. Consult your analysis program documentation for specific shape limitations of that program.

6.2.1 Shear Panel Element

Description

A plane element that only resists shear forces, tangential forces applied to the element edges. Some analysis programs also allow this element to resist normal forces through the use of effectiveness factors.

Application

Representing structures which contain very thin elastic sheets, typically supported by stiffeners.

Shape

Planar, three-noded triangle, four-noded quadrilateral, six-noded triangle, eight-noded quadrilateral. Some shapes are not available for all analysis programs.

Element Coordinate System

Refer to the figure in Section 6.2, "Plane Elements". The material angle can be used to rotate the element X axis.

Properties

Thickness, Nonstructural mass/area, Effectiveness Factors (not supported by many analysis programs).

Formulation

None.

6.2.2 Membrane Element

Description

A plane element that only resists in-plane normal (membrane) forces. In some analysis programs this is a degenerate form of the general plate element.

Application

Used to represent very thin elastic sheets.

Shape

Planar, three-noded triangle, four-noded quadrilateral, six-noded triangle, eight-noded quadrilateral. Some shapes are not available for all analysis programs.

Element Coordinate System

Refer to the figure in Section 6.2, "Plane Elements". The material angle can be used to rotate the element X axis.

Properties

Thickness, Nonstructural mass/area.

Formulation

DYNA option between Standard or Fully Integrated Belytschko-Tsay Membrane. ABAQUS option for Standard or Reduced Integration Membrane. This formulation has no affect on MARC.

6.2.3 Bending Only Element

Description

A plane element that resists only bending forces. In some analysis programs this is a degenerate form of the general plate element.

Application

Used to model plates that will only resist bending.

Shape

Planar, three-noded triangle, four-noded quadrilateral, six-noded triangle, eight-noded quadrilateral. Some shapes are not available for all analysis programs.

Element Coordinate System

Refer to the figure in Section 6.2, "Plane Elements". The material angle can be used to rotate the element X axis.

Properties

Thickness, Nonstructural mass/area, Bending Stiffness parameter (Nastran only), Fiber distances for stress recovery.

Formulation

None.

6.2.4 Plate Element

Description

A combined planar shell element. This element typically resists membrane (in-plane), shear, and bending forces. Some analysis programs also include transverse (through the thickness of the element) capabilities.

Application

Any structure which is comprised of thin plates/shells.

Shape

Planar, three-noded triangle, four-noded quadrilateral, six-noded triangle, eight-noded quadrilateral. Some shapes are not available for all analysis programs.

Element Coordinate System

Refer to the figure in Section 6.2, "Plane Elements". The material angle can be used to rotate the element X axis.

Properties

Thickness (average, or varying at each corner), Nonstructural mass/area, Bending Stiffness parameter (Nastran only), Transverse shear/Membrane thickness (Nastran only), Bending, Shear and Membrane-Bending Coupling Materials (Nastran only), Fiber distances for stress recovery.

Additional Notes

Many analysis programs do not support tapered plate elements. For those that do, you can specify a different thickness for each corner of the plate. You can always specify a single thickness for all corners simply by entering the average thickness.

Plate Offsets (Nastran, ANSYS, LS-DYNA Only) can be defined to offset the plate a particular distance from its nodes. Only one offset may be specified, and it will be in the plate's positive or negative normal direction.

Formulation

DYNA choice of 20 different element formulations. User selection is written to the SECTION_SHELL card.

No MARC options are available for this element type.

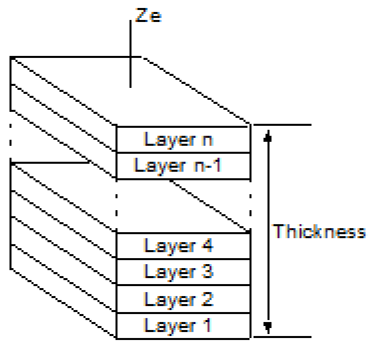
ANSYS Plate options for None - Ignore (no offset - SHELL63, with offset - SHELL181, KEYOPT (3) = 2), Standard (SHELL181, KEYOPT (3) = 2), Reduced Integration (SHELL181, KEYOPT (3) = 0). Thin Shell is not valid, therefore defaults to options used for None - Ignore.

ABAQUS Plate options for Standard (S3, S4, STRI65, S8R), Reduced Integration (S3R, S4R, S8R5, S8R), or Thin shells (STRI35, S4R5, STRI65, S8R5) can be defined. In addition, you can select Flat Triangles to export STRI3 elements instead of STRI35. The Warping option is only applicable to ABAQUS EXPLICIT, which causes S4RSW elements to be written instead of S4RS elements.

6.2.5 Laminate Element

Description

Similar to the plate element, except that this element is composed of one or more layers (lamina). Each layer can represent a different material. To create a laminate you need a *Layup* to specify the material, thickness, orientation angle and global ply ID (optional) of each ply and a *Laminate* property.



Application

Usually used to represent laminated composite shells.

Shape

Planar, three-noded triangle, four-noded quadrilateral, six-noded triangle, eight- noded quadrilateral. Some shapes are not available for all analysis programs.

Element Coordinate System

Refer to the figure in Section 6.2, "Plane Elements". The material angle can be used to rotate the element X axis. In addition, the material axes of each layer can be rotated in the element XY plane, relative to the element X axis.

Properties

For each laminate property, a Layup (a separate FEMAP entity containing material, orientation angle, thickness, and global ply ID for each ply) must be created and referenced by the property. Also, Bottom Surface, Nonstructural mass/area, Options, Bond Shear Allowable, and a Failure Theory. Not all options are available for all analysis programs

Formulation

DYNA choice between ten different element formulations. User selection is written to the SECTION_SHELL card. No MARC options are available for this element type.

ABAQUS Plate options for Standard (S3, S4, STRI65, S8R), Reduced Integration (S3R, S4R, S8R5, S8R), or Thin shells (STRI35, S4R5, STRI65, S8R5) can be defined. In addition, you can select Flat Triangles to export STRI3 elements instead of STRI35. The Warping option is only applicable to ABAQUS EXPLICIT, which causes S4RSW elements to be written instead of S4RS elements.

6.2.6 Plane Strain Element

Description

This is a biaxial plane element. It creates a two-dimensional model of a solid structure which does not vary through its depth, the plane strain condition.

Note: Some analysis programs use two-dimensional elements for plane strain analysis. Those programs usually require that the elements be located in a specific global plane. The required planes for each program are given in Section 6.3.1, "Axisymmetric Element". See also Section 8, "Analysis Program Interfaces" and your analysis program documentation for more information about which programs use two dimensional elements. You must build your model in one of the listed planes if you plan to use one of these analysis programs.

Application

Modeling of very thick solids which have one constant cross section.

Shape

Drawn as planar, but really a volume. Triangles represent wedges, quadrilaterals represent hexahedra. Three-noded triangle, four-noded quadrilateral, six-noded triangle, eight-noded quadrilateral. Some shapes are not available for all analysis programs.

Element Coordinate System

Refer to the figure in Section 6.2, "Plane Elements". The material angle can be used to rotate the element X axis.

Properties

Thickness and fiber distances (often not needed), Nonstructural mass/area.

Formulation

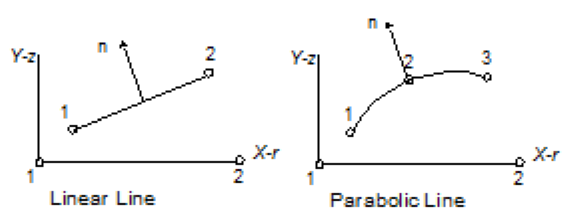
DYNA choice between Plane Strain or Plane Stress elements. You can also select Plane Stress for ANSYS by checking *Plane Stress* under the ABAQUS/MARC and ANSYS options area. The default is Plane Strain.

NX Nastran offers specialized Plane Strain and Plane Stress elements for Advanced Nonlinear (SOL 601) only. Set the NASTRAN option to “1” for Plane Strain or “2” for Plane Stress.

For ABAQUS and MARC, the elements will be written as Plane Strain unless the *Plane Stress* option is selected. The following table provides the elements associated with the different options. These elements correspond to linear and parabolic triangular and quadrilateral topologies. Certain options may only effect specific element topologies.

Analysis Program--> Plane-->	ABAQUS Strain	ABAQUS Stress	MARC Strain	MARC Stress
Standard	CPE3, CPE4, CPE6, CPE8	CPS3, CPS4 CPS6, CPS8	6, 11, 125, 27	3, 124, 26
Reduced Integration	CPE3, CPE4R, CPE6, CPE8R	CPS3, CPS4R, CPS6, CPS8R	6, 115, 125, 54 or 58	114,53
Incompatible Modes	CPE3, CPE4I	CPS3, CPS4I	Standard	3 (Assumed Strain)
Modified Contact	CPE6M	CPS6M	No effect	No effect
Hybrid	Add “H”	Add “H”	6, 11, 128, 32 or 58	No effect

6.2.7 Axisymmetric Shell Element



Description

This element is line element used to represent shells of revolution.

Application

Modeling of axisymmetric shell structures with axisymmetric constraints and loading, deforming in the radial plane.

Shape

Linear and parabolic lines defined by 2 or 3 nodes.

Element Coordinate System

Element orientation for Abaqus: (SAX1, SAX2)

For Abaqus these elements must be modeled in the XY plane with the r-direction positive. The r-direction is aligned with global X-direction and the z-direction corresponds to the global Y-direction.

The “top” surface of the shell is defined as the positive normal direction from node 1 to 2 of the loaded element. Pressure Loads can be defined on the “top” or bottom surface of the shell

See the figure above (in this section).

Properties

Thickness

Formulation

None

6.2.8 Plot Only Element (Plane)

Description

This element is nonstructural. It does not add any stiffness to your model. It is only used for plotting. Planar plot-only elements are also used by the ABAQUS interface to create interface elements and rigid bodies.

Application

Used to represent structural features that are not being analyzed, but that aid in the visualization of the model.

Shape

Linear quadrilateral and linear triangular shapes are allowed (midside nodes cannot be created) connecting 3 or 4 nodes.

Element Coordinate System

None

Properties

None

Formulation

None

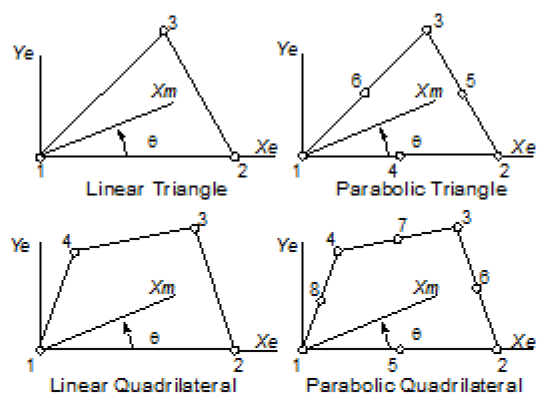
6.3 Volume Elements

These elements are all used to model three-dimensional solid structures. They can provide very detailed results, but usually require additional modeling and analysis time and effort.

6.3.1 Axisymmetric Element

Description

The axisymmetric element is a two-dimensional element used to represent volumes of revolution.



Note: Before using axisymmetric elements, it is very important to consult your analysis documentation. Most analysis programs require you to construct your model in a specific global plane.

The following table lists the required conventions for the supported programs:

Program	Global Model Plane	Radial Direction
NASTRAN	XZ	Global X
ANSYS	XY	Global X
ABAQUS	XY	Global X
MARC	XY	Global Y
LS-DYNA	XY	Global X
FEMAP Structural	XZ	Global X

The following table lists the required conventions for the obsolete programs:

Program	Global Model Plane	Radial Direction
MSC/pal & CDA/Sprint	XZ	Global X
STARDYNE	XY	Global X
COSMOS	XY	Global X
ALGOR, mTAB & SAP	YZ	Global Y
WECAN	XY	Global X

If possible, you should always build your model in the convention of the program you plan to use. To properly translate your model to any of the programs, FEMAP requires that you build the model using one of the listed conventions. When you write your model using one of the program translators, FEMAP checks to see if the model is in the correct plane for that program. If it is not, you will be given several options to automatically rotate it into the correct plane. If you do not use one of the above conventions, FEMAP will translate your model, but the results may be incorrect.

Application

Modeling of axisymmetric solid structures with axisymmetric constraints and loading.

Shape

Drawn as planar, but really represent axisymmetric rings. Three-noded triangle, four-noded quadrilateral, six-noded triangle, eight-noded quadrilateral. Some shapes are not available for all analysis programs.

Element Coordinate System

See the figure above (in this section). The material angle can be used to rotate the element X axis. Note the differences between the axisymmetric element coordinate angles and those for the plane elements. In this case, the angles are from a global axis, not from the first side of the element.

Properties

None.

Formulation

For Nastran, there are three options:” 0..Default”, “1..CTRIAX6, CTRAIX, CQUADX”, and “2..CTRAX3, CQUADX4, CTRAX6, CQUADX8”. When “0..Default” is selected, NX Nastran will use “2..CTRAX3, CQUADX4, CTRAX6, CQUADX8”, while all other Nastrans will use “1..CTRIAX6, CTRAIX, CQUADX”. You can choose the “1..CTRIAX6, CTRAIX, CQUADX” formulation when exporting for NX Nastran, but setting the formulation to “2..CTRAX3, CQUADX4, CTRAX6, CQUADX8” will cause an error in all other versions of Nastran, as these element types do not exist.

For DYNA you can choose between an Area or Volume Weighted formulation.

Both ABAQUS and DYNA have typical axisymmetric elements (2-DOF) as well as axisymmetric elements with twist. The 2-DOF elements will be used unless the *Twist* option is selected. The following table provides the elements associated with the different options. These elements correspond to linear and parabolic triangular and quadrilateral topologies. Certain options may only affect specific element topologies

.

Analysis Program--> Type-->	ABAQUS 2-DOF	ABAQUS Twist	MARC 2-DOF	MARC Twist
Standard	CAX3, CAX4 CAX6, CAX8	CGAX3, CGAX4 CGAX6, CGAX8	2, 10 126, 28	20 67
Reduced Integration	CAX3, CAX4R CAX6, CAX8R	CGAX3, CGAX4R CGAX6, CGAX8R	2, 116, 126, 55	Standard
Incompatible Modes	CAX3, CAX4I	Standard	Standard	Standard
Modified Contact	CAX6M"	No effect	No effect	No effect
Hybrid	Add "H"	Add "H"	2, 10 126, 33	66
Hybrid+Reduced (MARC)			2, 116 129, 59	66

6.3.2 Solid Element

Description

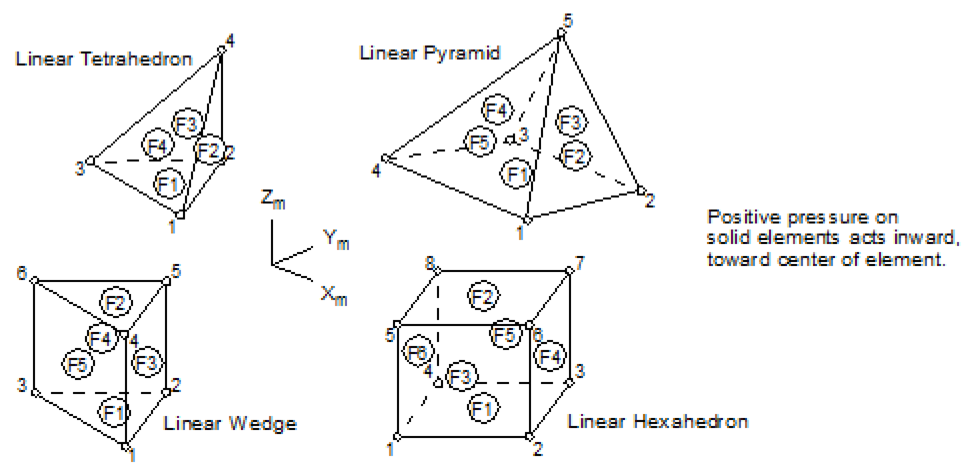
A three-dimensional solid element.

Application

Modeling of any three-dimensional structure.

Shape

Four-noded tetrahedron, five-noded pyramid, six-noded wedge, eight-noded brick (hexahedron), ten-noded tetrahedron, thirteen- noded pyramid, fifteen-noded wedge, and twenty-noded brick. Some shapes are not available for all analysis programs.



Element Coordinate System

Can be aligned based on the node locations or aligned to a coordinate system. Check your analysis program documentation for supported options.

Properties

Material axes, integration order (not all programs).

Formulation

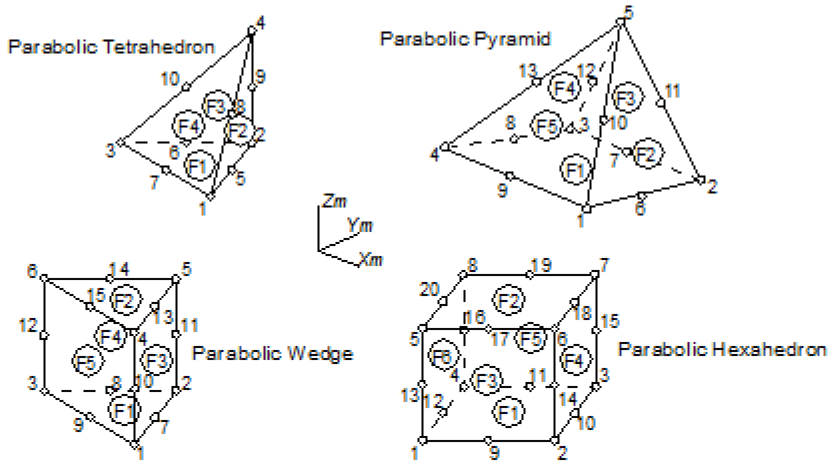
For DYNA you can choose between 17 element formulations, although you will typically want to choose one of these for 4-noded tetrahedrals: (1) 1-Stress Point, (2) Fully Integrated S/R, (3) Fully Integrated 6-DOF/Node, (4) Tetrahedral 6-DOF/Node or (10) 1-Point Tetrahedron (default). For 10-noded tetrahedrals: (1) 1-Stress Point, (16) 10 Node Tetrahedron, or (17) 10 Node Composite Tetrahedron (Default). The default is (18) EQ -1: Fully Integ S/R for Poor Aspect Ratio, Efficiency. The selected option is output to the *SECTION_SOLID card.

Both ABAQUS and DYNA have element metric elements with twist. The 2-DOF elements will be used unless the Twist option is selected. The following table provides the elements associated with the different options. These elements correspond to linear and parabolic triangular and quadrilateral topologies. Certain options may only affect specific element topologies.

Analysis Program-->	ABAQUS	MARC
Standard	C3D4, C3D6, CRD8 C3D10, C3D15, C3D20	134, 7 127, 21
Reduced Integration	C3D4, C3D6, C3D8R C3D10, C3D15, C3D20R	134, 117 127, 57
Incompatible Modes	C3D4, C3D6, C3D8I	7 (Assumed Strain)
Modified Contact	C3D10M	No effect
Hybrid	Add “H”	2, 10 126, 33
Hybrid+Reduced (MARC)		2, 116 129, 59

Additional Notes

If you want to apply pressure loads to solid elements, you must specify a face number. The previous and following figures, show the face numbers (F1 through F6, in the circles) for each element shape. Positive pressure is always directed inward, toward the center of the element.



Linear and Parabolic Pyramid elements are only supported for NX Nastran and MSC Nastran.

6.3.3 Solid Laminate Element

Description

Similar to three-dimensional solid element, except that this element is composed of one or more layers (lamina). Each layer can represent a different material. To create a solid laminate you need a *Layup* to specify the material, thickness, orientation angle and global ply ID of each ply and a *Solid Laminate* property.

Application

Usually used to represent laminated composites using a single layer of solid elements through the thickness.

Shape

Six-noded wedge, eight-noded brick (hexahedron), fifteen-noded wedge, and twenty-noded brick. Some shapes are not available for all analysis programs.

Element Coordinate System

Aligned to a material coordinate system where a ply/stack direction is also selected (one axis is designated as the ply orientation direction, while another is designated as the stacking direction). Check your analysis program documentation for supported options.

Properties

For each solid laminate property, a Layup (a separate FEMAP entity containing material, orientation angle, thickness, and global ply ID for each ply) must be created and referenced by the property. Also, Material Coordinate System, Ply/Stack Direction, Bond Shear Stress Allowable, Bond Normal Stress Allowable for Failure Theory. In addition, a Ply Failure Theory, a Bond Failure Theory and the required values for each may be specified on the Ply/Bond tab of several material types. Not all options are available for all analysis programs.

Formulation

None

6.4 Other Elements

This category of elements allows you to define masses, rigid connections, general stiffnesses, contact pairs and slide lines.

6.4.1 Mass Element

Description

A generalized three-dimensional mass and/or inertia element located at a node. The center of mass can be offset from the node. An even more general form is the mass matrix element.

Application

Representing parts of a structure which contain mass, but which do not add any stiffness.

Shape

Point, connected to one node. Symbol is a square separated into 4 smaller squares (2 shaded, 2 unshaded).

Element Coordinate System

Aligned with a coordinate system that you specify. Some analysis programs require that you define masses relative to global rectangular coordinates or the nodal degrees of freedom.

Properties

Mass (or MassX, MassY, and MassZ for some programs), Inertias (Ixx, Iyy, Izz, Ixy, Iyz, Izx), Offsets.

Formulation

None

6.4.2 Mass Matrix Element

Description

A generalized three-dimensional mass and/or inertia element. The mass and inertia properties are defined as a 6x6 mass matrix. In most cases, the mass element is easier to define.

Application

Representing parts of a structure which contain mass, but which do not add any stiffness.

Shape

Point, connected to one node. Symbol is a square separated into 4 smaller squares (2 shaded, 2 unshaded) inside [ ].

Element Coordinate System

Aligned with a coordinate system that you specify. Some analysis programs require that you define masses relative to global rectangular coordinates or the nodal degrees of freedom.

Properties

Upper triangular portion of a 6x6 mass matrix.

Formulation

None

6.4.3 Spring/Damper to Ground Element

Description

A combined stiffness (spring) and damper element to ground. The DOF spring to ground is an alternative element.

Application

Used to specify a bushing element which is "grounded"

Shape

Has a circular symbol with a line beneath the symbol, at a single node.

Element Coordinate System

If *Orientation* is set to *CSys* on the element itself, then the element Csys is equal to the selected Csys. If instead, *Orientation* is set to *From Property* on the element AND *Orientation CSys* on the property is enabled, then the element Csys is equal to the selected Csys on the property.

Properties

Stiffness, Damping, and Structural Damping values can be defined for individual degrees of freedom, along with Spring/Damper Location (can also be specified on element), Orientation Csys (can also be specified on element), Stress/Strain recovery coefficients. For Frequency or nonlinear analysis function dependence can be defined for stiffness and damping values.

Formulation

None

6.4.4 DOF Spring to Ground Element

Description

A combined stiffness (spring) and damper element. This element can use any (of six) nodal degree of freedom at the only node. The spring/damper to ground element is an alternative element to create a bushing.

Application

Used to connect a degree of freedom with "ground", having a specified stiffness.

Shape

Drawn as a "jagged" symbol with a line beneath the symbol, at a single node.

Element Coordinate System

Determined by *Connect to DOF* degree of freedom.

Properties

Degree of Freedom (at single node), stiffness, damping.

Formulation

There are 2 formulations available for NX Nastran and MSC.Nastran. "0..Default (CELAS2/CDAMP2)" will write a CELAS2 or CDAMP2 which have both "property" (i.e. stiffness value in the "K" field for CELAS2 or damping value in the "B" field for CDAMP2) and "connection" (i.e., node/grid ID) information in a single Nastran entry. When "1..CELAS1/CDAMP1" is chosen, a CELAS1 or CDAMP1 will reference an appropriate Property (PID) for a spring (PELAS) or damper (PDMAP). The PELAS and PDAMP are not written at all when using the default formulation.

6.4.5 Rigid Element

Description

Represents a rigid connection between a master node and one or more other nodes. FEMAP has no limit on the number of additional nodes, or the degrees of freedom which may be connected on these additional nodes. Weighting factors for these connections may also be defined.

Some analysis programs require that the rigid element connects all six degrees of freedom. Other programs let you limit the connection to selected degrees of freedom. In addition, support for the rigid element weighting factors in analysis programs is limited.

Application

Modeling connections which are very stiff relative to the remainder of the structure.

Shape

One master node, connected to one, to nineteen, additional nodes. If element formulation for Nastran is set to 1..RSPLINE then the element will have at least two independent nodes and at lest 1 dependent node.

Element Coordinate System

None, depends on nodal degrees of freedom.

Properties

None.

Formulation

Rigid element formulations are only currently supported for Nastran, ABAQUS, MSC.Marc, and ANSYS. These formulations allow you to export FEMAP rigid elements as Nastran RSPLINEs, ABAQUS MPC type, both methods for using MPC184 as a “Link/Beam” for ANSYS, or MSC.Marc TYING command.

NX Nastran or MSC.Nastran

There are 2 formulations available for Nastran. The default “0..Default” defines RBE2 or RBE3 elements. If the formulation is set to “1..RSPLINE” then FEMAP will define the RSPLINE element.

ABAQUS

There are currently thirteen different ABAQUS *MPC types supported by FEMAP. The following table provides a list of the different supported MPCs, the number of required slave nodes, whether the order of the slave nodes are important, the total number of MPCs written for each rigid element, and the output format. If any of these options are unclear, please cross-reference this table with the Multi-Point Constraints section in the ABAQUS/Standard users manual. The references to Node a, b, S, etc. are taken directly from this manual.

MPC Type	No. of Slave Nodes/ Order	No. of MPCs	Output Format/Comments
BEAM, TIE, PIN LINK, ELBOW	>0/Unimportant	No. Slave Nodes	Same master (2nd node in ABAQUS MPC) for all MPCs but different slave node.
SS LINEAR SS BILINEAR SSF BILINEAR	>1/Important	Single	Master node as Shell Node, S. Slave nodes exported as Solid Nodes, p values.
SS BEAM	>2/Important	Single	First 2 slave nodes (s1, s2) written as Pipe Axis. Master then written as beam node, b. Remaining slaves (s3, etc.) written after master.
REVOLUTE	2/Important	Single	Master as Node a. First Slave as Node b. Second slave as Node c to define rotation.
SLIDER	2/Important	Single	Master node as Sliding Node, p First 2 Slave Nodes as sliding axis, a, b.
CYCLSYM	3/Important	Single	Master as First Edge Node a First Slave as Second Edge Node b. Second and Third Slaves as cyl symmetry axis c, d
UNIVERSAL	3/Important	Single	Master as First Node a. First Slave as Node b. Second and Third Slaves (c, d) define rotation.
V LOCAL	3/Important	Single	Master as Velocity Constrained Node a. First Slave defines local rotation direction (Node b) Second Slave defines velocity (Node c)

ANSYS

There are 3 formulations available for ANSYS. The default is “0..ANSYS CP/CERIG” which creates CP (set of coupled degrees of freedom) or CERIG (rigid region) elements depending on what is selected during export of the analysis model.

The other two formulations create MPC184 elements (link/beam type elements only). You can choose to use the "Direct Elimination" method for imposing kinematic constraints (1..MPC184 Direct Elimination) or the "Lagrange Multiplier" method (2..MPC184 Lagrange Multiplier). Elements must be 2-noded rigid elements only in FEMAP (1 independent node to 1 dependent node).

DOFs	Type of MPC184 KEYPOT(1)	Kinematic Method KEYOPT(2)	Use CTE?
TX, TY, TZ	= 0, Rigid Link Element	= 0, Direct Elimination	No
TX, TY, TZ	= 0, Rigid Link Element	= 1, Lagrange Multiplier	Yes
TX, TY, TZ RX, RY, RZ	= 1, Rigid Beam Element	= 0, Direct Elimination	No
TX, TY, TZ RX, RY, RZ	= 1, Rigid Beam Element	= 1, Lagrange Multiplier	Yes

MSC.Marc

Currently there are three TYING options supported. If formulation is set to *0..None* then Femap will write MPC types 1-6 if specific degrees of freedom are selected or type 100 if all six dof are selected.

If formulation is set to *Marc MPC Type 80* then type 80 will be used for the TYING option.

MSC.Marc MPC Type	No. of Slave Nodes	No. of MPCs	Output Format/Comments
1-6	>0	No. Slave Nodes	Same master for all MPCs but different slave node.
100	>1	Single	Dependent (slave) node is the node to be tied. Independent (master) node is written as retained node.
80	>1	Single	Dependant (slave node) written as tied node. Independent (Master) written as retained node. A third Extra node not connected to the structure is created automatically to account for the rotational dof.

6.4.6 General Matrix Element

Description

A general matrix element which can be used to define a stiffness matrix, damping matrix, or mass matrix. This element lets you define a 12 x 12 matrix or a 6x6 stiffness matrix that will be symmetrically applied (expanded to a 12x12 matrix) to two nodes.

Application

Modeling of custom stiffness, damping, or mass connections between two nodes that cannot be adequately represented by other available element types.

Shape

Drawn as a line with a symbol which is "X" inside [ ]. Real shape is undefined. Only symbol will appear if nodes are coincident.

Element Coordinate System

Depends on nodal degrees of freedom.

Properties

Select a *Coordinate System*, a *Matrix Type*, and enter values into the upper triangular portion of a 12x12 or 6x6 stiffness matrix.

Formulation

None

6.4.7 Slide Line Element

Description

A contact element which allows input of frictional and stiffness contact information between nodes on surfaces. Input includes a series of master and slave nodes to define geometry of surface contact.

Application

Modeling of finite-sliding surface interaction between two deformable bodies.

Shape

Drawn as lines between master and slave nodes.

Element Coordinate System

None. Depends on coordinate system of nodes.

Properties

Width of contact surfaces and stiffness/frictional data including Stiffness Scale Factor, Nonsliding Frictional Stiffness, and Static Friction Coefficient.

6.4.8 Weld/Fastener Element

Description

A weld element allows you to connect "entities" (Elements, Nodes, or patches of Nodes) which connect the nodes of the "entities" with an element of a specific diameter, material, and assigned orientation. A fastener element uses a similar type of connection, though the fastener has fewer definition options. There a quite a few more options on the corresponding property for a fastener such as mass, structural damping, and material coordinate system to go along with a specified diameter.

Application

Simulates a spot weld between two groups of finite element "entities" (Elements, Nodes, or patches of Nodes).

Shape

Drawn as a lines connecting the nodes of each group of finite element "entities" (Elements, Nodes, or patches of Nodes) to the spot where the weld/fastener orientation intersects the "entity". The two sets of lines are then connected using a "tube" the size of the assigned diameter.

Note: If a weld endpoint is a single Element Vertex, (i.e., a node) then the "tube" will go from that Element Vertex to the lines or another Element Vertex, depending on the type of weld specified.

Element Coordinate System

By default, for both welds and fasteners, this depends on coordinate system of nodes. A separate material coordinate system can be specified for fasteners, which can be absolute or relative.

Properties

For Weld, Diameter of weld connection, Spot Weld designation (Yes/No), and whether to Eliminate M-Set DOF or not.

For Fastener, Diameter of the faster connection, Translational Stiffness values (KTX, KTY, and KTZ), Rotational Stiffness values (KRX, KRY, and KRZ), Material Coordinate System in which Translational and Rotational Stiffness are applied, whether the Material Coordinate System is Absolute or Relative, Mass of fastener, and Structural Damping.

7 Translation Tables for Analysis Programs

This topic defines the entities that the FEMAP interfaces transfer to finite element analysis programs. Two translation tables list the entities for these FEMAP interfaces:

- Section 7.1, "Translation Table for ANSYS, I-DEAS, NASTRAN, and MSC Patran"
- Section 7.2, "Translation Table for ABAQUS, LS-DYNA, and MSC.Marc"

See also:

- Section 8, "Analysis Program Interfaces"

Using the Translation Tables

The following pages contain tables that show how FEMAP entities are translated to and from the supported analysis programs.

The numbered notes in the tables refer to the notes that follow the tables. You will find the notes are numbered from 1-N for each analysis program. Therefore, if you see note 6 in the column for ANSYS Write translator, that refers to Note 6 for ANSYS, not note 6 for one of the other programs.

You will see numerous <— symbols listed for the various read translators. Wherever you see this symbol, it means that the read translator supports the same entities as the corresponding write translator. The column for the write translator is to the left of the column for the read translator, hence the <— symbol. In some cases, the read translator will support more entities than the write translator. In those cases you will see "<— +" followed by the additional entities that are supported. Whenever an entity is not supported by the corresponding read or write translator (or by the analysis program), you will see "---"

7.1 Translation Table for ANSYS, I-DEAS, NASTRAN, and MSC Patran

FEMAP	ANSYS		I-DEAS		NASTRAN		MSC Patran	
	Write	Read	Write	Read	Write	Read	Write	Read
Coordinate Systems								
Rectangular	LOCAL (KCS=0)[6]	<——	Dataset 2420[1]	<——	CORD2R[1]	<—+ CORD1R	Type 5[1]	<——
Cylindrical	LOCAL (KCS=1)[6]	<——	Dataset 2420[1]	<——	CORD2C[1]	<— + CORD1C	Type 5[1]	<——
Spherical	LOCAL (KCS=2)[6]	<——	Dataset 2420[1]	<——	CORD2S[1]	<— + CORD1S	Type 5[1]	<——
Nodes								
All	N, NROTAT[1]	<——+ NMODIF[1]	Dataset 2411	<——	GRID SPOINT EPOINT	<——	Type1[2]	<——
Elements								
Rod	EN (LINK8)[29] LINK10)[32]	<—— + E[12]	Dataset 2412 (fe 11)	<——	CROD	<—— + CONROD	Packet 2[3] (Cfg=3)	<——
Bar	EN (BEAM4)[2,23]	<——+ E[12]	Dataset 748, 2412 (fe 21)	<—— +(2438)	CBAR[40]	<——	Packet 2[3] (Cfg=1)	<——
Tube	EN (PIPE16)	<—— + E[12]	Dataset 2412 (fe 21)	<——	CTUBE (not UAI, Cosmic)	<——	---	---
Curved Tube	EN (PIPE18)	<—— + E[12]	Dataset 748, 2412 (fe 23)	<—— +(2438)	CBEND (not UAI, Cosmic)	<——	---	---
Link	---	---	---	---	---	---	---	---
Beam	EN (BEAM44[2], BEAM188 ko[3]=3)[41]	<—— + E[12]	Dataset 748, 2412 (fe 22)	<—— +(2438)	CBEAM[40] (Cosmic-CBAR)	<——	Packet 2[3] (Cfg=2)	<——
Spring/Damper	EN (COMBIN14) ko[3]=0,1 EN (MATRIX27) ko[1]=0 ko[2]=0 ko[3]=4,5	<——+ E[12]	Dataset 2412 (fe 136, 137)	<——	CROD[3], CVISC, CBUSH[25]	as Rod, CVISC, CBUSH[25]	---	---
DOF Spring	EN (COMBIN14) ko[2]=1-6[7]	<—— + E[12]	---	---	CELAS1, CELAS2, CDAMP1, CDAMP2[4]	<—— + CELAS1, CDAMP1[23]	Packet 2[3] (Cfg=6)	<——
Curved Beam	---	---	Dataset 748, 2412 (fe 23)	<—— +(2438)	CBEND[5](not UAI, Cosmic)	<——	---	---
Gap	EN (CONTAC52) ko[1]=1	<—— + E[12]	---	---	CGAP (not Cosmic)	<——	---	---
Plot Only	KNODE, L	---	Dataset 2431	<——	PLOTEL	<——	---	---
Shear	EN (SHELL28) quad only[3]	<—— + E[12]	---	---	CSHEAR quad only	<——	Packet 2[3] (Cfg=7)	<——
Parabolic Shear	---	---	---	---	---	---	Packet 2[3] (Cfg=7)	<——
Membrane	EN (SHELL63) ko[1]=1[9]	<——+E[12] + (PLANE42) ko[3]=0,3	Dataset 748, 2412 (fe 41, 44)	<——	CTRIA3, CQUAD4, CQUADR, CTRIAR	<——+ CTRMEM, CQDMEM	Packet 2[3] (Cfg=5)	<——
Parabolic Membrane	---	EN + E[12] + (PLANE82) ko[3]=0,3	Dataset 748, 2412 (fe 42, 45)	<—— +(2438)	CTRIA6, CQUAD8[22] (not Cosmic)	<——	Packet 2[3] (Cfg=5)	<——
Bending	EN (SHELL63) ko[1]=2[9]	<—— + E[12]	---	---	CTRIA3, CQUAD4, CQUADR, CTRIAR	<—— + CTRPLT, CQDPLT	Packet 2[3] (Cfg=10)	<——
Parabolic Bend-ing	---	---	---	---	CTRIA6, CQUAD8[22], (not Cosmic)	<——	Packet 2[3] (Cfg=10)	<——
Plate	EN (SHELL63, SHELL57, stif43) ko[1]=0[9]	<—— + E[12]	Dataset 748, 2412 (fe 91, 94)	<—— +(2438)	CTRIA3, CQUAD4, CQUADR, CTRIAR	<—— + CTRIA1, CQUAD1, CTRIA2, CQUAD2	Packet 2[3] (Cfg=4)	<——
Parabolic Plate	EN (SHELL93)[9]	<—— + E[12]	Dataset 748, 2412 (fe 92, 95)	<—— +(2438)	CTRIA6, CQUAD8[22] (not Cosmic)	<——	Packet 2[3] (Cfg=4)	<——

FEMAP	ANSYS		I-DEAS		NASTRAN		MSC Patran	
	Write	Read	Write	Read	Write	Read	Write	Read
Plane Strain	EN (PLANE42, PLANE55, stif56) ko[3]=2[3]	<—— + E[12] (do not read any hyper)	Dataset 748, 2412 (fe 51, 54)	<—— +(2438)	CTRIA3, CQUAD4, CQUADR, CTRIAR (not UAI, Cosmic) CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8 CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8[52]	<——	Packet 2[3] (Cfg=11)	<——
Parabolic Plane Strain	EN (PLANE82, PLANE77, stif74) ko[3]=2[3]	<—— + E, +(stif2)[12]	Dataset 748, 2412 (fe 52, 55)	<—— +(2438)	CTRIA6, CQUAD8[22] (not UAI, Cosmic)	<——	Packet 2[3] (Cfg=11)	<——
Laminated Plate	EN (stif53) ko[3]=NL tri only[8,9]	<—— + E[12]	---	---	CTRIA3, CQUAD4, CQUADR, CTRIAR	<——	---	---
Parabolic Laminated Plate	EN(SHELL91, SHELL99) ko[3]=NL[9, 33]	<—— + E[12]	---	---	CTRIA6, CQUAD8[22] (not Cosmic)	<——	---	---
Planar Plot Only	---	---	Dataset 2431	<——	---	---	---	---
Axisymmetric	EN (PLANE42, PLANE55, stif56) ko[3]=1[3]	<—— + E[12]	Dataset 748, 2412 (fe 81, 84)	<—— +(2438)	CQUADX4[38] CQUADX8[38] CTRAX3[38] CTRAX6[38] CTRIAX6, CTRIARG, CTRIAX, CQUADX[7]	<——	Packet 2[3] (Cfg=8)	<——
Parabolic Axisymmetric	EN (PLANE82, PLANE77, stif74) ko[3]=1[3]	<—— + E, +(stif2)[12]	Dataset 748, 2412 (fe82,85)	<—— +(2438)	CTRIAX6, CTRIAX, CQUADX[7] (not UAI, Cosmic)	<——	Packet 2[3] (Cfg=6)	<——
Solid	EN (SOLID45 or stif73, SOLID70, stif58), ko[4]=1[22]	<—— + E[12] + (stif72)	Dataset 748, 2412 (fe 111, 112, 115)	<—— +[2](2438)	CTETRA, CPYRAM[67] CPENTA, CHEXA (Cosmic-CTETRA, CWEDGE, CHEXA1, CHEXA2, CIHEX1)	<—— + CHEXA1, CHEXA2, CWEDGE	Packet 2[3] (Cfg=0)	<——
Parabolic Solid	EN (SOLID95, SOLID87, SOL-ID90, SOLID92) EMORE[22]	<—— + E[12]+	Dataset 748, 2412 (fe 118, 113, 116)	<—— +[2](2438)	CTETRA, CPENTA, CHEXA (Cosmic-CIHEX2)	<—— + CHEX20	Packet 2[3] (Cfg=0)	<——
Solid Laminate	---	---	---	---	CPENTA, CHEXA	<——	---	---
Parabolic Solid Laminate	---	---	---	---	CPENTA, CHEXA	<——	---	---
Mass	EN (MASS21) ko[3]=0[10]	<—— + E[12,13]	Dataset 2412 (fe 161)	<——	CONM2[8]	<——	Packet 2[3] (Cfg=7)	<——
Mass Matrix	EN (MATRIX27) ko[3]=2[20]	<—— + E[2,20]	---	---	CONM1	<——	---	---
Spring/Damper to Ground	EN (COMBIN14) ko[3]=0,1 EN (MATRIX27) ko[1]=0 ko[2]=0 ko[3]=4,5	<——+ E[12]	---	---	CBUSH[25]	<——	---	---
DOF Spring to Ground	EN (COMBIN14) ko[2]=1-6[7]	<—— + E[12]	---	---	CELAS1, CELAS2, CDAMP1, CDAMP2[4]	<—— + CELAS1, CDAMP1[23]	Packet 2[3] (Cfg=6)	<——
Rigid	EN(mpc184)[38] ko[1]=0 or 1 ko[2]=0 or 1[39] CP, CEAVE, CERIG[24]	<——[14]	Dataset 748, 2412 (fe 122)	<—— +(2438)	RBE1, RBE2, RBE3[41], (CRBE2, CRBE3 -Cos-mic) RSPLINE[26]	<—— + RBAR[2] (CRBAR-Cosmic)	---	Packet 14[6]
General Matrix	EN (MATRIX27) ko[3]=2,4,5[11]	<—— + E[11,12]	---	---	---	---	---	---
Slide Line	---	---	---	---	BLSEG, BCONP	---	---	---
Weld/Fastener	---	---	---	---	CWELD, CFAST	---	---	---

FEMAP	ANSYS		I-DEAS		NASTRAN		MSC Patran	
	Write	Read	Write	Read	Write	Read	Write	Read
Properties								
Rod	R (1)[29]	<——[15]	Dataset 776, 2437	<—— +[3](789)	PROD	<——	Packet 4[4]	<——
Bar	R (1-6,8- 10,12) RMODIF	<— + R(7)[15]	Dataset 776, 2437	<—— +[3](789)	PBAR PBARL[42]	<——	Packet 4[4]	<——
Tube	R (1,2)	<——[15]	Dataset 776, 2437	<—— +[3](789)	PTUBE[10]	<——	---	---
Curved Tube	R (1-3)	<——[15]	Dataset 776, 2437	<—— +[3](789)	PBEND	<——	---	---
Link	---	---	---	---	---	---	---	---
Beam	R (1- 24,55) RMODIF SECTYPE[40] SECDATA[40] SECOFFSET[40] SECCONTROLS[40] SECNUM[40]	<—— + R(53)[15]	Dataset 776, 2437	<—— +[3](789)	PBEAM[11] PBEAML[42]	<——	Packet 4[4]	<——
Spring/Damper	R (1,2) RMODIF(1-78)	<——[15]	Dataset 2437	<—— +[3](789)	PROD[3], PVISC, PBUSH, PBUSHT[25]	as Rod, PVISC, PBUSH, PBUSHT[25]	---	---
DOF Spring	R (1,2)	<——[15]	---	---	on Elem PELAST PELAS[47] PDAMP[47] PDAMPT	<——	Packet 4[4]	<——
Curved Beam	---	---	Dataset 776, 2437	<—— +[3](789)	PBEND[5]	<——	---	---
Gap	R (1,2), MU[4]	R (1,2)[4,15]	---	---	PGAP	<——	---	---
Plot Only	---	---	---	---	---	---	---	---
Shear	R (1)	<—— + ESYS[15]	---	---	PSHEAR	<——	Packet 4[4]	<——
Parabolic Shear	---	---	---	---	---	---	Packet 4[4]	<——
Membrane	R (1,5)	<—— + ESYS[15]	Dataset 2437	<—— +[3](789)	PSHELL[12]	<—— +PTRMEM, PQDMEM, PQDMEM1, PQDMEM2	Packet 4[4]	<——
Parabolic Membrane	---	---	Dataset 2437	<—— +[3](789)	PSHELL[12]	<——	Packet 4[4]	<——
Bending	R (1,5)	<—— + ESYS[15]	Dataset 2437	<—— +[3](789)	PSHELL[12]	<—— + PTRPLT, PQDPLT	Packet 4[4]	<——
Parabolic Bend- ing	---	---	Dataset 2437	<—— +[3](789)	PSHELL[12]	<——	Packet 4[4]	<——
Plate	R (1-5) SECTYPE[40] SECDATA[40] SECOFFSET[40] SECCONTROLS[40] SECNUM[40]	<—— + ESYS[15]	Dataset 2437	<—— +[3](789)	PSHELL[12]	<—— + PTRIA1, PQUAD1, PTRIA2, PQUAD2	Packet 4[4]	<——
Parabolic Plate	R (1-4,6) SECTYPE[40] SECDATA[40] SECOFFSET[40] SECCONTROLS[40] SECNUM[40]	<—— + ESYS[15]	Dataset 2437	<—— +[3](789)	PSHELL[12]	<——	Packet 4[4]	<——
Plane Strain	none	ESYS[15]	Dataset 2437	<—— +[3](789)	PSHELL, PLPLANE[12] PPLANE[61]	<——	Packet 4[4]	<——
Parabolic Plane Strain	none	ESYS[15]	Dataset 2437	<—— +[3](789)	PSHELL, PLPLANE[12] PPLANE[61]	<——	Packet 4[4]	<——
Laminated Plate	R (1-60)[37]	<—+ ESYS[15]	---	---	PCOMP[13,29] PCOMPG[46]	<——	---	---
Parabolic Laminated Plate	R (1-3,7- 9, 13-15 ...)[21,37]	<—— + ESYS[15,21]	---	---	PCOMP[13,29] PCOMPG[46]	<——	---	---
Axisymmetric	none	ESYS[15]	Dataset 2437	<—— +[3](789)	PLPLANE	<——	Packet 4[4]	<——
Parabolic Axisymmetric	none	ESYS[15]	Dataset 2437	<—— +[3](789)	PLPLANE	<——	Packet 4[4]	<——
Solid	ESYS	<——[15]	Dataset 2437	<—— +[3](789)	PSOLID[14] (+PIHEX- Cosmic), PLSOLID	<——	Packet 4[4]	<——

FEMAP	ANSYS		I-DEAS		NASTRAN		MSC Patran	
	Write	Read	Write	Read	Write	Read	Write	Read
Parabolic Solid	ESYS	<——[15]	Dataset 2437	<—— +[3](789)	PSOLID[14] (+PIHEX- Cosmic), PLSOLID	<——	Packet 4[4]	<——
Solid Laminate	---	---	---	---	PCOMPS[65]	<——	---	---
Parabolic Solid Laminate	---	---	---	---	PCOMPS[65]	<——	---	---
Mass	R (1-6)	<——[15]	Dataset 2437	<—— +[3](789)	on Elem	<——	Packet 4[4]	<——
Mass Matrix	R (1-78)[20]	R (1-6, 13-17, 24-27, 34-36 43,44,51)[15,20]	---	---	on Elem	<——	---	---
Spring/Damper to Ground	R (1,2) R(1-78) RMODIF[42]	<——[15]	Dataset 2437	<—— +[3](789)	PROD[3], PVISC, PBUSH, PBUSHT[25]	as Rod, PVISC, PBUSH, PBUSHT[25]	---	---
DOF Spring to Ground	R (1,2)	<——[15]	---	---	on Elem PELAST PELAS[47] PDAMP[47] PDAMPT	<——	Packet 4[4]	<——
Rigid	none	none	none	none	none	none	---	---
General Matrix	R (1-78) RMODIF[11, 42]	<——[11,15]	Dataset 2437	<—— +[3](789)	---	---	---	---
Slide Line	---	---	---	---	BFRIC, BOUTPUT, BWIDTH	---	---	---
Weld/Fastener	---	---	---	---	PWELD, PFAST	---	---	---
Materials								
Isotropic	EX, GXY, NUXY, DENS, ALPX, DAMP, TREF, REFT(R5), KXX, C, MPTEMP, TB(BKIN, MKIN, MISO, BISO, DP, MELAS)	<—— + MPDATA, MP[16]	Dataset 1710	<——[4]	MAT1, MAT4, MATT1, MATT4, TABLEM1[28] TABLEM2[28] CREEP, MATS1, TABLES1, TABLEST MATFT[66]	<—— + TABLEM1 TABLEM2 TABLEM3	Packet 3	<——
Orthotropic -2D	EX, EY, GXY, NUXY, DENS, ALPX, ALPY, DAMP, TREF, REFT(R5), KXX, KYY, C, MPTEMP[30]	<—— + MPDATA, MP[16,30]	Dataset 1710	<——[4]	MAT8, MAT5, MATT8, MATT5, TABLEM1,[28] TABLEM2[28]	<— + TABLEM1, TABLEM3	Packet 3	<——
Orthotropic -3D	EX, EY, EZ, GXY, GYZ, GXZ, NUXY, NUYZ, NUXZ, DENS, ALPX, ALPY, ALPZ DAMP, TREF, REFT(R5), KXX, KYY, KZZ, C, MPTEMP[30]	<—— + MPDATA, MP[16,30]	Dataset 1710	<——[4]	MAT3, MAT5, MATT3, MATT5. TABLEM1[28] TABLEM2[28] MAT11, MATT11[48] MAT12, MATT12[49] MATFT[66]	<——[9]+ TABLEM1, TABLEM3	Packet 3	<——
Anisotropic -2D	EX, EY, GXY, NUXY, DENS, ALPX, ALPY, DAMP, TREF, REFT(R5), KXX, KYY, C, TB(BKIN, MKIN, MISO, BISO, DP, MELAS), MPTEMP[5,30] TB [34,36]	<——	Dataset 1710	<——[4]	MAT2, MAT5, MATT2, MATT5, TABLEM1[28] TABLEM2[28] CREEP, MATS1, TABLES1, TABLEST	<—— + TABLEM1, TABLEM3	Packet 3	<——

FEMAP	ANSYS		I-DEAS		NASTRAN		MSC Patran	
	Write	**Read**	**Write**	**Read**	**Write**	**Read**	**Write**	**Read**
Anisotropic -3D	EX, EY, EZ, GXY, GYZ, GXZ, NUXY, NUYZ, NUXZ, DENS, ALPX, ALPY, ALPZ DAMP,TREF, REFT(R5), KXX, KYY, KZZ, C, TB(BKIN, MKIN, MISO, BISO, DP, MELAS), MPTEMP[5,30] TB [34,36]	<——	Dataset 1710	<—— [4]	MAT9, MAT5, MATT9, MATT5, TABLEM1[28] TABLEM2[28], CREEP, MATS1, TABLES1, TABLEST MATFT[66] (Cosmic MAT6)	<—— + TABLEM1, TABLEM3	Packet 3	<——
Hyperelastic	---	---	---	---	MATHP	---	---	---
Other	---	---	---	---	MATHE[30] MATHEM[58] MATHEV[59] MATVE[60] MATG[34] MAT10[37] MATMSA[56] NITONAL[57]	---	---	---
Functions								
All	---	---	---	---	---	---	---	---
Constraints								
Constraint Combination	---	---	---	---	SPCADD, MPCADD[43]	<——	---	---
Nodal	D (=0.), M	<——[17,18]	Dataset 791	<—— [5]	SPC, ASET, BSET, CSET, QSET, OMIT, SUPORT1,SUPORT[16]	<—— + SPC1,ASET1, BSET1,CSET1, QSET1,OMIT1,	Packet 8[5]	<——
Nodal NonZero	D (not 0.)	<——[17,18]	Dataset 791	<—— [5]	SPC1	<——	Packet 8	<——
Equation	CE	<——[19]	Dataset 754	<—— [6]	MPC[17]	<——	---	---
Nodal Loads								
Load Combination	---	---	---	---	LOAD[44]	<——	---	---
Force and Moment	F	<——[18]	Dataset 790	<—— [7]	FORCE, MOMENT, SLOAD[27]	<—— + FORCE1, MOMENT1	Packet 7	<——
Displacement	---	---[17,18]	---	---	SPCD[18] TIC	<——	Packet 8	<——
Velocity	---	---	---	---	TIC	---	---	---
Acceleration	---	---	---	---	ACCEL1[70]	<——	---	---
Temperature	T(R4), BF(R5)[25]	<——	Dataset 792	<—— [7]	TEMP, Heat Transfer SPC, TEMPBC	<——	---	---
Heat Generation	BF(HGEN)	<——	---	---	SLOAD	<——	Packet 10	<——
Heat Flux	F(HEAT)	<——	---	---	QHBDY	<——	---	---
Nonlinear Transient	---	---	---	---	NOLIN1 thru 4, TF	---	---	---
Elemental Loads								
Distributed Load	SFBEAM	<——	---	---	PLOAD1	<——	---	---
Pressure	EP(R4), SFE(R5)[26]	<——	Dataset 790	<—— [7]	PLOAD, PLOAD2, PLOAD3, PLOAD4, PLOADX1, PLOADE1[63]	<——	Packet 6	<——
Temperature	TE(R4), BFE(R5)[27]	<——	---	---	TEMPRB, TEMPP1[19]	<——	Packet 6	<——
Heat Generation	BFE(HGEN)	<—	---	---	QVOL	<——	Packet 11	<——
Heat Flux	SFE(HFLUX)	---	---	---	QBDY1	<——	---	---
Convection	SFE(CONV)	---	---	---	CONV, CONVM, PCONV, PCONVM	CONV PCONV	---	---

FEMAP	ANSYS		I-DEAS		NASTRAN		MSC Patran	
	Write	Read	Write	Read	Write	Read	Write	Read
Radiation	---	---	---	---	RADBC, RADM, RADMT, VIEW, VIEW3D, RADCAV, RADSET	<——	---	---
Body Loads								
Translational Acceleration	ACEL	<——	Dataset 790	<—— [8]	GRAV	<——	---	---
Rotational Acceleration	DCGOMG	<——	Dataset 790	<——	RFORCE[21]	<——	---	---
Rotational Velocity	CGOMGA	<——	Dataset 790	<——	RFORCE[21]	<——	---	---
Rotation Origin	CGLOC	<——	Dataset 790	<——	RFORCE	<——	---	---
Varying Translational Acceleration	---	---	---	---	ACCEL[64]	<——	---	---
Default Temperature	TUNIF	<—— + BFUNIF, TEMP(R5)	Dataset 792	<——	TEMPD	<——	---	---
Heat Transfer	---	---	---	---	PARAM,TABS and SIGMA	---	---	---
Aeroelasticity								
Aero Panel/Body	---	---	---	---	CAERO1, CAERO2	<——	---	---
Aero Property	---	---	---	---	PAERO1, PAERO2	<——	---	---
Aero Splines	---	---	---	---	SPLINE1, SPLINE2, SET1	<——	---	---
Aero Control Surfaces	---	---	---	---	AESURF, AELIST	<——	---	---
Static Aero	---	---	---	---	AEROS, AESTAT, TRIM, AEROF, APRES PARAM,AUNITS	<——	---	---
Flutter	---	---	---	---	AERO, MKAERO2, FLUTTER, FLFACT, FMETHOD	<—— MKAERO1	---	---
Miscellaneous								
CONTACT	RMODIF CONTA171, CONTA172, CONTA173, CONTA174, TARGE169, TARGE170, ko[2,4-9, 11-12][31]	RMODIF CONTA173 CONTA174 TARGE170	Dataset 164		BSURF[31] BCTSET[31] BCTADD[31] BCTPARM[31] BCTPARA[31] BCRPARA[31] BCPROP[31] BCPROPS[31] BGSET[31] BGADD[31] BGPARM[31] NXSTRAT[31] BCONTACT[71] BCBODY[71] BCTABLE[71] BCPROP[71] BCPARA[71] NLPARM[71] BSSEG[24] BSCONP BLSEG[39,50]	<——	---	---
Groups	NSEL, ESEL, CM	<——	---	---	---	---	---	---
Fluid Region	---	---	---	---	MFLUID[53]	<——	---	---
Bolt Region	---	---	---	---	BOLT[32]	<——	---	---
Bolt Preload	---	---	---	---	BOLTFOR[32]	<——	---	---
NonStructural Mass Region	---	---	---	---	NSM1, NSML1, NSMADD[67]	<—— NSM, NSML	---	---
Rotor Region	PRETS179	---	---	---	ROTORG[33]	<——	---	---
	CSYS, MAT, REAL, ET, KEYOPT, TYPE, ESYS[28]	<—— + NUMSTR (elem only)			CHBDYP, CHBDYG, PHBDY, TLOAD1, DLOAD, LOAD[20]	---	---	---
	Rev 4 only: RSIZE, CPSIZE, CESIZE				EIGR, EIGRL, EIGC EXTRACTMETHOD[62]	<——	---	---

FEMAP	ANSYS		I-DEAS		NASTRAN		MSC Patran	
	Write	Read	Write	Read	Write	Read	Write	Read
Case Control for Dynamics and Nonlinear Analysis	---		---	---	DLOAD, TSTEP, TSTEPNL, NLPARM, NONLINEAR, SDAMPING, FREQUENCY, RANDOM,	<——	---	---
	Rev 4: LWRITE, TEDELE, TDELE, FDELE, EPDELE, DDELE, CEDELE Rev 5: BFEDELE,BFDELE, FDELE, SFEDELE,DDELE, CEDELE <—Rev4: LWRITE Rev5: LSWRITE, SOLVE				PARAM - AUTOSPC, GRDPNT, WTMASS, K6ROT, MAXRATIO, NDAMP, INREL[54], LANGLE, LGDISP, LGSTRN, PRGPST, OGEOM, AUTOMPC[35] , DDRAM, MODACC, RESVEC, RESVINER RIGIDELEM2ELAS, RIGIDELEMTYPE[55] SWPANGLE[73] MGRID[72], MDOF[72] NOFISR[35] SRCOMPS, BOLTFACT[35] ENFMOTN[51] BAILOUT SECOMB APLHA1, ALPHA2	<—— WTMASS[6]	---	---
					Other PARAM[36] - COUPMASS, RSPECTRA, W3, W4, CURVPLOT, KDAMP, FZERO, WMODAL, LMODES, LFREQ, HFREQ, G, SCRSPEC, OPTION (ABS, SRSS, NRL, NRL0) CLOSE, ROTGPF, ROTCSV, MULTBC, BODYLOAD, GPFORCE, MPCFORCE, PELMCHK, NOOUT, NOSTR, NOELF, NODIS, NOLOD, NOSPC, DSA, OPTIM, OLDSEQ, NMLOOP, LGSTRN, OPTION(CQC)[45]	COUPMASS, RSPECTRA, W3, W4, KDAMP LMODES, FZERO, WMODAL, LFREQ, HFREQ, G, SCRSPEC, OPTION (ABS, SRSS, NRL, NRL0) CLOSE, NMLOOP, LGSTRN, OPTION(CQC)[45]	---	---
						GRDSET, BAROR, BEAMOR	---	---
	/PREP7, /NOPR, /GOPR, /VIEW, /ANGLE, TOTAL, FINISH Rev 4: KAN, ITER, AFWRITE Rev 5: ANTYPE, OUTPR, OUTRES		---	---	TSTEP, TSTEPNL, NLPARM, NCPCI, FREQ, FREQ1, FREQ2, FREQ3, FREQ4, FREQ5, RLOAD2, TLOAD1, DPHASE, LSEQ, RANDPS, RANDT1, DTI, TABDMP1, WEIGHTCHECK, GEOMCHECK, GROUNDCHECK, OMODES	TSTEP, TSTEPNL, NLPARM, NLPCI, FREQ, FREQ1, FREQ2, FREQ3, FREQ4, FREQ5, RLOAD2, TLOAD1, RANDPS, RANDT1, DTI, TABDMP1, GEOMCHECK, WEIGHTCHECK, GROUNDCHECK	---	---
External Superelement Creation	---		---	---	EXTSEOUT, ASSIGN, OUTPUT2 ASSIGN, OUTPUT4	<——	---	---
External Superelement Use	---		---	---	ASSIGN, INPUTT2 UNIT PARAM,SECOMB	<——	---	---
NASTRAN Line commands and System Cells (NX Nastran)	---	---	---	---	SYSTEM(442) SYSTEM(319) ITER, ELEMITER PARALLEL ENFMOTN[51]	ITER, ELEMITER ENFMOTN[51]	---	---
Iterative Solver (MSC Nastran)	---		---	---	SMETHOD[69]	<——	---	---
	TB [35]	---	---	---	---		---	---

FEMAP	ANSYS		I-DEAS		NASTRAN		MSC Patran	
	Write	Read	Write	Read	Write	Read	Write	Read
Case Control and Bulk Data Delimiter cards	---	---	---	---	SUBCASE SUBCOM[68] BEGIN BULK ENDDATA	<——	---	---

7.1.1 ANSYS Translation Notes

1. FEMAP defines the directions of nodal degrees of freedom using an output coordinate system. At each node, the directions of the nodal degrees of freedom in the output coordinate system are used to calculate nodal rotations (on N or NROTAT) for ANSYS. FEMAP supports either the ANSYS Rev 4 or Rev 5 angle definition. FEMAP permanent constraints are merged with the nodal constraints in each constraint set translated.

2. For beam (STIF44) elements, Keyopts 3 and 4 are used to represent the FEMAP beam releases in ANSYS Rev 4. Only the elemental rotational degrees of freedom can be released. In ANSYS Rev 5, Keyopts 7 and 8 are used and any combination of DOF can be released. Releases on translational DOF will generate a warning. Releases on bar elements (STIF4) are not supported. These elements should be converted to beams, or if possible modify the properties to eliminate the stiffness in the released direction. For Rev 5, FEMAP will translate nonstructural mass to and from the additional mass real constant.

3. FEMAP orients the material coordinate system for planar elements using a specified angle for each element. Certain elements in ANSYS do not readily support this approach. For the shear (STIF28), plane strain and axisymmetric (STIF42, STIF82 and STIF2) elements, FEMAP does not support elemental material angles. For these elements, you must enter all material properties in global coordinates. ANSYS does not recommend using triangular STIF42 elements. You should convert to triangular STIF82 (parabolic) elements, or use the linear triangles with great care.

4. When FEMAP writes a GAP (STIF52) property which has a coefficient of friction a MU command is written. Only the Y-direction coefficient of friction is used. You will receive an error message if you specify a different coefficient of friction in the Y and Z directions. The MU command is not supported by the read translator and all friction will be ignored. FEMAP will write compression stiffness, gap distance, and transverse stiffness for gap elements. You will receive an error message if you specify a nonzero tension stiffness since it is not supported by STIF52. The starting condition is not supported and is left blank. It will be skipped by the read translator.

5. When anisotropic materials are translated to ANSYS, the full matrix representation that you input is lost. The matrix input is converted to an effective orthotropic representation prior to translating. You should review this approximation carefully.

6. FEMAP does not support elliptical coordinate systems, or the elliptical factors on the LOCAL command. FEMAP does support the differences in angle definitions between ANSYS Rev 4 and Rev 5. ANSYS coordinate systems must be numbered between 11 and 40 for ANSYS Rev 4, or 11 and 999999 for ANSYS Rev 5. Other coordinate systems are not supported for translation. Also, note the difference between ANSYS spherical coordinate systems and FEMAP spherical coordinate systems. The angular coordinates, and therefore, the degrees of freedom, are switched, and reoriented. For all entities which FEMAP handles in spherical coordinates, the differences are automatically corrected for you, but the switches can lead to confusion. It is often best to avoid spherical coordinates, if possible.

7. DOF spring elements in ANSYS (STIF14) must connect to the same DOF at both nodes.

8. This element is not recommended. It is only supported in ANSYS Rev 4, it has been dropped from Rev 5. Use the parabolic laminated plates (STIF91) instead.

9. FEMAP supports material angle definition for each element but ANSYS defines these angles with real constants. A model with an irregular mesh (many different angles) will have to generate many sets of real constants. You should use this capability with caution. For laminate plates, any material orientation angle specified for the element is combined with the angles specified for each layer when the real constants are defined.

10. When you translate for ANSYS Rev 4, mass elements must be defined in global coordinates unless they are specified as a pure lumped mass. That is, mass in all directions is the same, and there is no inertia. If specified, mass offsets and products of inertia are skipped. For ANSYS Rev 5, they can also be defined in the output coordinate system of the node where they are located.

11. FEMAP's general matrix element allows entry of a 12x12 matrix or a 6x6 matrix. The 6x6 matrix is automatically expanded to a 12x12 during translation to the STIF27 real constants. The expansion however does not take into account the geometric transformations required for non-coincident nodes. You will therefore receive a warning for each stiffness matrix element using only the 6x6 matrix that you translate which connects non-coincident nodes. When reading these elements, you will receive an error message if the 12x12 real constants are not equivalent to the internal 6x6 representation.

12. FEMAP reads the normal EN command format as well as the E command format. The NUMSTR,ELEM command is also supported to define the element ID for the E command. In addition to these normal PREP7 commands, FEMAP also supports a variation of the EN command that is generated by the CDWRITE command. The format of this variation is:

```
EN,4.4,      1,      1,      1,      1,      1,      0,      0
   1      2      3      4      5      6      7      8      9     10
  11     12     13     14     15     16     17     18     19     20
```

where the first line is EN, ANSYS Version, Order number (not read), MATL, ETYPE, REAL, Element ID, ESYS, and Select Key (not read). The next two lines contain up to 20 nodes which define the element. Unused nodes are 0.

13. Only 3D mass elements (KEYOPT 3 = 0 or 2) are read.

14. FEMAP rigid elements only support up to 20 nodes per element. If you have more nodes in a coupled DOF set, the set will be broken into multiple rigid elements. This limitation no longer exists and should be reviewed in a future release.

15. In addition to the R commands that FEMAP writes, The RMODIF and RMORE commands are also supported when reading an ANSYS model. In addition, if you set a material direction via the ESYS command, it sets the element orientation in FEMAP.

16. FEMAP can read the various material constant commands that it writes and also the MPDATA command (1st term only) which is generated by CDWRIT. FEMAP can also read the constant term from the MP command.

17. When reading D (displacement) commands, FEMAP creates constraints if the displacement value is 0.0, or NonZero constraints if value is non-zero.

18. When reading loads and constraints (D, M, F), FEMAP only reads the first fields which define the node, degree of freedom, and value. The other fields which specify additional DOF, IDs, or other data are simply skipped without generating warnings.

19. FEMAP can only read constraint equations (CE) which reference 20 or fewer degrees of freedom. Entering a negative node ID to remove nodes from the equation is not supported.

20. FEMAP mass matrix elements only allow specification of mass at one node, and are therefore represented as a 6x6 matrix. ANSYS STIF27 (KEYOPT 3=2) mass matrices connect to two nodes, and are therefore a 12x12. When FEMAP writes mass matrix elements, the second node is simply connected to the minimum node number in your model. Only the 6x6 real constants connect to the first node are entered with the values you specified - all others are entered as 0.0. Since the ANSYS Weight Generator looks at the center of the two nodes, this approach results in incorrect CG and inertia estimations from mass matrix elements, you will receive an error message if there is any nonzero mass or inertia associated with the second node point. In fact the second node is completely skipped and the mass is only connected to the first node.

21. FEMAP does not support writing or reading tapered laminate elements. Only a single thickness is allowed for each ply.

22. FEMAP always writes the STIF45 ET command with KEYOPT(4)=1. That is the material axes aligned with the element IJ nodes. Then, if your model contains properties that align the material axes to a specific coordinate system, FEMAP writes ESYS commands to override this specification. If you align with global rectangular coordinates (CSys 0), FEMAP will create a local coordinate system at the origin which is aligned to the Global axes and use it for the orientation. STIF73 and STIF95 elements do not support material axes which are aligned to the element IJ side, so you will receive an error message if you attempt to translate elements using this convention.

23. FEMAP always writes beam (STIF44) and bar (STIF4) elements with a third node to define the orientation. When reading these elements however, either the third node, or angular orientation method can be used.

24. FEMAP supports translating rigid elements to either CP or CERIG commands. If you have rigid elements in your model, FEMAP will ask which method you prefer to use in the translation. Normally CERIG commands are preferred since they generate the proper equations for rotational coupling, rather than just connecting specified DOF. In ANSYS Rev 4, however, they can only represent fully coupled (all 6 DOF) or translationally coupled (all 3 translational DOF) connections. If you need other connections, you must either use CP commands, or use constraint equations instead of rigid elements. In ANSYS Rev 5, CERIG commands support any combination of rigid DOF.

25. FEMAP uses T commands when translating nodal temperatures to ANSYS Rev 4. BF,,TEMP commands are used for Rev 5.

26. FEMAP uses EP commands when translating pressures to ANSYS Rev 4 and SFE,,PRES commands for Rev 5. Varying pressures at different corners of an element are not supported. Similarly, pressures on the edges of planar elements are not supported.

27. FEMAP uses TE commands when translating elemental temperatures to ANSYS Rev 4. BFE,,TEMP commands are used for Rev 5. FEMAP only supports one constant temperature for each element. In Rev 4, FEMAP translates the proper number of identical temperatures to define a uniform element temperature. In Rev 5, only one temperature is translated, since other temperatures will all default to the first, and automatically produce uniform element temperature.

28. In ANSYS Rev 4, element types were selected with a numeric value (e.g. 44 for STIF44 beams). In Rev 5, this changes to a combined name and numeric (e.g. BEAM44 for STIF44 beams). FEMAP writes the appropriate method for each version and can read either format.

29. Any rod torsional properties will be lost when translating to ANSYS STIF4 elements. STIF4s do not have any torsional stiffness.

30. The Poisson's ratio coefficients NUXY, NUYZ, and NUXZ follow a different convention than used by most other programs and FEMAP. The values that you enter are therefore converted to the ANSYS convention when you write an ANSYS file, and converted to the FEMAP convention when you read an ANSYS file. This involves the following conversion: NUij(ANSYS) = NUij(FEMAP) * Ej / Ei.

31. Surface-to-surface contact is defined in ANSYS using a combination of contact surface and target segment elements. ANSYS looks for contact only between contact surfaces and target segments which share the same real constant set. The real constant set is defined using the *Connect, Connection Property* command. By clicking the *ANSYS* tab, you can modify the KEYOPTs.

32. Supports both the tension only and compression only KEYOPT 3. FEMAP uses the CABLE property; For the Compression-only (Gap) option, turn the Compression Only Gap option on and set the *Initial Tension value* >= 0.0. For the Tension-only (Cable) option turn the Compression Only Gap option off and set the *Initial Slack value* <= 0.0.

33. FEMAP 8.1 and greater supports reading and writing of bottom surface offset for Analyses defined using the Analysis Case Manager only. Bottom surface offset is entered on the laminate property and is defined as the distance from the bottom of the element to location where the reference plane (or nodes) are located. A bottom surface value of (-0.5*t) or ("0.0" the default) will offset the nodes to the middle surface of the element, for nodes at the bottom surface enter a value 1E-15, and for nodes at the top surface enter (-t).

34. When exporting to ANSYS using the Analysis Case Manager, FEMAP will write the 2D and 3D Anisotropic material definition using the TB command. Anisotropic materials are supported for shell elements 181,182,183 and solid elements 185,186. When writing the TB card with the ANEL argument for Anisotropic materials, FEMAP does not support TBOPT option 1 (inverted stiffness matrix). When exporting an analysis file without using the Case Manager, FEMAP will write an effective orthotropic representation (See note 5).

35. Supports reading of the TB command, but only for the ANEL argument needed to read Anisotropic materials.

36. ANSYS requires only ALPX, ALPY, ALPZ for thermal expansion. FEMAP will only use the first three coefficients in the FEMAP definition material definition. ANSYS requires only KXX, KYY, KZZ for thermal conductivity. FEMAP will use the first row of the FEMAP material definition.

37. The "Options" in the *Laminate Definition* portion of the dialog box other than "1..Symmetric" are not supported.

38. MPC184 elements can only be specified with all translational DOFs checked or all DOFs checked. If this is not specified in the rigid element, you will get an error during translation, but the element will be written out as having all translational DOFs by default (KEYOPT(1)=0).

39. If you want the Coefficient of Thermal Expansion (CTE) specified for a rigid element in FEMAP to be taken into account during analysis, you must set the formulation for the rigid element to "2..MPC 184 Lagrange Multiplier". This will write KEYOPT(2)=1 to the ANSYS file for the corresponding MPC184 element type.

40. "SEC" entries only used for beams with formulations set to "1..BEAM188/Section Shape" or "2..BEAM188/ASEC" and linear and parabolic plate elements with offsets.

41. When beam element formulation is set to "1..BEAM188/Section Shape" or "2..BEAM188/ASEC", KEYOPT(3)=3 is always written to specify cubic shape function (KEYOPT, ‘element type ID’, 3, 3).

42. For *Spring/Damper* elements with *Type* set to *CBUSH*, *Stiffness* values for MATRIX27 written with KEYOPT(2)=4, while *Damping* values written for MATRIX27 with KEYOPT(2)=5. For *Spring/Damper to Ground* elements, an additional node is created, constrained in all 6 DOF, then used as node 2 of the MATRIX27. For *General Matrix* elements, *Matrix Type* set to “0..Stiffness” writes MATRIX27 with KEYOPT(2)=4, *Matrix Type* set to “1..Damping” writes MATRIX27 with KEYOPT(2)=5, and *Matrix Type* set to “2..Mass” writes MATRIX27 with KEYOPT(2)=2.

7.1.2 I-DEAS Translation Notes

1. FEMAP writes all coordinate systems, including the global systems, to the I-DEAS file. The global systems are renumbered, but are titled “FEMAP Global ...". If you read a universal file with these titles, FEMAP remaps them back to the global. Likewise, if you read a universal file that contains CSys 1 or 2, they are renumbered to valid FEMAP coordinate system IDs.

2. FEMAP can write, but not read, solid element material orientation vectors.

3. FEMAP supports a limited number of structural element properties for all element types. Any other properties that are written in the file will be skipped without warning.

4. Only I-DEAS constant material properties are supported. If your file contains other types of materials, or material constants that FEMAP does not support, FEMAP will still attempt to read universal file dataset 1710.

5. Constant zero displacement constraints are supported as normal nodal constraints in FEMAP. All non-zero values for constraints will be written and read assuming global rectangular coordinate system. Temperature and time varying constraints will be skipped.

6. Only constraint equations with up to 13 terms can be read. In addition, the constant term must be zero. Only real terms are read.

7. Time varying loads are not supported; they will be read as constants. Shell temperatures will be read as a single nodal value using the top temperature. FEMAP only reads the loading universal datasets if they correspond to one of the available FEMAP load types.

8. Gravity loads are skipped. Only the translational acceleration loads are read.

9. FEMAP only supports IDs up to eight digits. Any entities with IDs greater than eight digits will be lost when reading a universal file.

10. Beam neutral axis offsets from the shear center are not supported when translating data through the universal file. Property values are correct, but element offsets are not computed to reattach the nodes correctly. In addition, curved beam properties do not support neutral axis offsets from the shear center. If you have any of these conditions, transfer data via a Nastran file.

11. If you export a complete part from I-DEAS, coordinate systems will be written in Dataset 1961, which is not supported. To properly transfer, simply export only the FE data.

12. If you export multiple FE models in the same universal file, only the first one will be read.

7.1.3 NASTRAN Translation Notes

The following translation notes apply to translations for NX Nastran, MSC Nastran, CSA/NASTRAN, UAI/NASTRAN, Cosmic/NASTRAN, ME/NASTRAN, Autodesk Nastran (NEi/Nastran), and SSS/NASTRAN. All programs use similar file formats.

1. Coordinate systems 1 and 2 are predefined in FEMAP. They are always written to your Nastran data file, but use a special continuation field that tells FEMAP's Nastran read translator to skip them. If you read a Nastran model which has a coordinate system 1 or 2 (and does not have the special continuation field) those coordinate systems, and all entities which reference them will be automatically renumbered.

2. FEMAP will read RBAR elements if all six independent elemental DOF are connected to one node. This is the typical case for a true rigid bar. If the model is then translated back to Nastran, the elements will be converted to RBE2 elements.

3. FEMAP spring elements are automatically converted to equivalent CROD elements during write. A representative PROD and MAT1 card are generated based on the element length and specified spring constants. Damping is not supported.

4. The stress coefficient, S, is not supported for CELAS2 elements. If you specify the spring stiffness as 0.0, FEMAP will write a CDAMP2 element instead of the CELAS2. To have FEMAP export CELAS1 and CDAMP1 entries and corresponding PELAS and PDAMP entries to the input file, the element formulation must be changed to “1..CELAS1/CDAMP1”.

5. Type 3 CBEND cards are written. Type 1, 2 or 3 can be read, but for type 1 or 2, the radius on the PBEND card must be 0. Only the primary CBEND format is supported. The ThetaB field is not supported on the PBEND card.

6. If you have a PARAM,WTMASS entry in your model, FEMAP will automatically multiply all material densities, mass, and inertias for CONM2 elements, and matrix entries for CONM1 elements by the WTMASS factor. When FEMAP writes a Nastran model, the WTMASS parameter is not written.

7. CQUADX elements are only written when a hyperelastic material is referenced by an Axisymmetric Property and the formulation is set to “1..CTRIAX6, CTRAIX, CQUADX” (NX Nastran and MSC Nastran) or “0..Default” (MSC Nastran). In addition, the nodes of the CQUADX must lie on the XY plane instead of the XZ plane, which is different from other Axisymmetric elements in Nastran. The user will be prompted to automatically move the nodes to the XY plane, should they not already be there, then set the proper elements normal direction.

8. Nastran only supports one mass component on CONM2. FEMAP uses the X-direction mass for this value on write, and sets all three components equal on read.

9. 3D Orthotropic materials that are used by axisymmetric elements are translated to MAT3 commands. Most data is directly translated for this command, but the NUXY term must be converted as it is translated to or from the NUZX MAT3 term. The conversion is required because NUZX would really correspond to NUYX, not NUXY. The MAT3 NUZX is set equal to NUXY * EY / EX.

10. FEMAP's tube element is not tapered, so the OD2 field is not supported.

11. FEMAP does support tapered beams, with properties at both ends, but does not support intermediate property data. In addition, FEMAP only supports shear center and neutral axis offsets. The additional nonstructural mass center of gravity offset is not supported.

12. PSHELL is used for all of the plate-like element properties. The various MID fields control the type of plate. For a plane strain representation MID2=-1. For a membrane MID2=0 and MID3=0. For a bending representation MID1 and MID2 are not 0, but MID3 is 0. For a full plate representation none of the MIDs are 0. FEMAP also supports full definition of all of the material IDs.

13. No Longer Valid - In FEMAP versions prior to 9.3, the number of laminate plies was restricted to 90 plies. This is no longer the case with the introduction of the Layup entity.

14. Only the material and integration network are supported for PSOLID.

15. No Longer a Limitation

16. The type of Nastran card written by the nodal constraints is controlled by your selections during the translation process. Any FEMAP constraint set that is not selected as one of the other sets will automatically be translated as SPC cards. FEMAP does not support nonzero displacements, which are defined on SPC cards.

17. FEMAP only supports MPC equations with up to 70 terms.

18. Nodes referenced on SPCD cards must also be selected by a nodal constraint. FEMAP does not do this automatically for you. You must create nodal constraints for all nodes using SPCDs.

19. FEMAP only supports gradient information (TPRIME) on the TEMPP1 entry. All other information on the TEMPP1 and TEMPRB entries (i.e., T1/T2 on TEMPP1 and TA/TB/TP on TEMPRB) are ignored.

20. LOAD cards are automatically generated if you combine body and nodal/elemental loads in the same FEMAP case. The LOAD card takes the ID of the FEMAP load case, and all of the actual loads are renumbered to nonexisting cases which are then combined by LOAD. Temperatures remain in the original case.

21. For MSC and UAI/Nastran, both tangential acceleration and rotational velocity are supported on the RFORCE command, but they must be along/around the same vector.

22. For parabolic plate elements, Nastran evaluates the material axis orientation at each interior integration point, along the constant parametric lines of the element. FEMAP orients, based on the edge going from nodes 1-2 on the element. As long as your elements have straight sides, with the midside nodes “at the midside”, these are equivalent. If not, the Nastran representation will not match what you defined in FEMAP. In fact, the Nastran formulation will point in different physical directions at each interior integration point.

23. FEMAP does not support spring/damper elements that refer to scalar points to indicate a constrained coordinate.

24. For NEi/Nastran, contact can be defined by creating *Connectors*. The *Connection Region* type must be set to *Deformable*. If the *Connection Region*s are defined using nodes then the Output must be set to *Nodes*. When *Connection Region*s are defined by *Elements* or *Surfaces* the output must be set to *Elements. Connection Region*s will be read from the input file and may be read as node lists or converted to element faces.

25. CBUSH elements are defined using a element formulation. There are two formulations for a Spring element. 0..Default defines CROD or CVISC elements. When the formulation is set to 1..CBUSH then the Nastran CBUSH element and corresponding PBUSH will be written to the input file. If the Orientation Csys defined in the property is ON then the element Csys is equal to the selected Csys. If the Orientation Csys is OFF then the element Csys is defined with the X axis going from the first node to the second. The element Y axis is perpendicular to the element X axis. It points from the first node toward the orientation (or third) node. If you use an orientation vector, the Y axis points from the first node in the direction of the orientation vector. The element Z axis is determined from the cross product of the element X and Y axes. Element offsets are only supported using the nodal output coordinate system and will be transformed when read into FEMAP.

26. The RSPLINE element is defined using a FEMAP Rigid element and setting the Element Formulation to “1..RSPLINE”. The First and last terms in the RSPLINE must be defined as independent.

27. SLOAD's are 1 dimensional concentrated loads. They can only be applied to SPOINTs in FEMAP by defining the X component of a Nodal Force. FEMAP will read SLOAD(s) as a Nodal Heat Generation load from a thermal analysis or a Nodal Force from all other solution types.

28. NX Nastran Advanced Nonlinear Analysis (SOL 601) does not support TABLEM2 entries for material function dependence. It does however support the use of TABLEM1 entries, which do NOT use specific field values on material entries (MAT1, MAT4, etc.) as multipliers. Therefore, all functionally-dependent material properties for SOL 601 should be defined with the actual values in the function and a “1” should be placed in those fields of the MAT* entry to produce the TABLEM1 entry. For TABLEM2, which is exported from FEMAP for many types of analysis, the values specified in the *General* tab of the material dialog box represent scalars, so it is recommended to also place the material values in the function directly and simply use the scalars as multipliers to these values.

29. FEMAP 9.1 and greater supports reading and writing of bottom surface offset for Analyses defined using the Analysis Case Manager only. Bottom surface offset is entered on the laminate property and is defined as the distance from the bottom of the element to location where the reference plane (or nodes) are located. A bottom surface value of (-0.5*t) or (“0.0” the default) will offset the nodes to the middle surface of the element, for nodes at the bottom surface enter a value 1E-14, and for nodes at the top surface enter (-t).

30. FEMAP 9.3 and greater supports reading and writing of the hyperelastic materials for NX Nastran Solutions 601 and 701 (MATHE). These materials are found in “Other Types” and the supported types are the Mooney-Rivlin, Hyperfoam, Ogden, Arruda-Boyce, an Sussman-Bathe. Also, the MATHEs for MD/MSC Nastran Solution 600 are supported.

31. Contact entities for linear contact, glued contact, and advanced nonlinear contact are only supported for NX Nastran.

32. Bolt Preload entities supported for NX Nastran only.

33. Rotor Dynamics entities supported for NX Nastran only.

34. The MATG is supported for NX Nastran Solution 601 only. Also, “Gasket Results” can only be reviewed in FEMAP by requesting results in the Nastran binary output file (.op2).

35. AUTOSPC (“1..Singular Value Decomposition” option), AUTOMPC, BOLTFACT, and NOFISR PARAMs are supported for NX Nastran Only.

36. These PARAMs are created by setting options in various commands of FEMAP for different analysis types and other options, not using the NASTRAN Bulk Data dialog box in the Analysis Set Manager

37. The MAT10 material is a Fluid Material and can only be used with solid elements.

38. These Axisymmetric element types are only available when using NX Nastran version 6.0 and above.

39. *Connection Regions* for 2-D contact in Solution 601 of NX Nastran (usually in conjunction with axisymmetric elements) must be defined using nodes only and are written out to the Nastran file as BLSEG entries. The nodes must be selected in proper order with contact occurring to the “left side” of the region. The BCTSET entry is used to specify which BLSEG entries are in contact with one another. If a BLSEG is specified as “Rigid”, it must be the “target” in the *Connector* (Contact Pair). BLSEG entries in Nastran input files for solution sequences other than Solution 601 represent “slideline” elements.

40. When a bar or beam property has been created in FEMAP using the “Shape” button to define the cross-section, the cross-section definition values will also be exported as a comment to the Nastran file. For example, the Height; Width, Top; Width, Bottom; Thickness, Top; Thickness, Bottom; and Thickness values would be exported for an I-Beam and appear in as a comment such as this: “$ Femap with NX Nastran Section 1 : 9,0,1.,0.5,0.75,0.1,0.1,0.05”. If a Nastran input file containing these type of comments is imported, the type of cross-section and corresponding values will also be entered into the “Cross Section Definition” dialog box and the beam/bar cross-sections can then be viewed graphically in your FEMAP model.

41. To use the CTE on rigid elements, you must check the “Rigid Element Thermal Expansion” option in the “Plate, Beam, and Rigid Options” section of the *NASTRAN Bulk Data Options* dialog box of the Analysis Set Manager (*Model, Analysis* command). Otherwise, the CTEs will be ignored.

42. When PBARL and PBEAML properties are imported into FEMAP from a Nastran input file, the beam property section values will be calculated in FEMAP using the same algorithm Nastran uses to evaluate PBARL and PBEMAL entries when solving. This is done for consistency and efficiency purposes.

43. SPCADD and MPCADD constraint combinations are available for all solution sequences except Steady-State and Transient Heat Transfer.

44. LOAD load combinations are available for Linear Static, Nonlinear Static, and Buckling Analysis.

45. Method available for NEi Nastran only

46. PCOMPG entries are only generated when all plies also have a Global Ply ID set. If any ply doesn’t have a Global Ply ID defined, then a PCOMP will be generated instead. When results are imported into FEMAP from a Nastran run which contained PCOMPG entries, the results ply-by-ply results will be associated with the Ply ID in FEMAP, not the Global Ply ID.

47. FEMAP can write PELAS, PELAST, PDAMP, and PDAMPT entries in combination with CELAS1 and CDAMP1 elements by setting the element formulation to “1..CELAS1/CDAMP1

48. For NX Nastran, FEMAP writes out MAT11 and MATT11 (if needed) bulk data entries when there are solid elements which use a 3-D Orthotropic material.

49. For NEi Nastran, FEMAP writes out MAT12 and MATT12 (if needed) bulk data entries when there are solid elements which use a 3-D Orthotropic material.

50. For “Edge-to-Surface” glued contact in NX Nastran, the *Connection Region* for the “edge region” must be defined with nodes from elements of certain topology (3 or 6-noded triangles, 4 or 8- noded quads) and set as the *Source* in the *Connector*. This will create a BLSEG entry, which is used to define the “edge region” that can be glued to the “face” of solid or shell elements. The shell or solid element face region (creates a BSURF, BSURFS, BCPROP, or BCPROPS entry) must be entered as the *Target* in the *Connector*.

51. Turning on ENFMOTN in the PARAM section of the NASTRAN Bulk Data Options writes out different things for NX Nastran and MSC/MD Nastran. For NX Nastran, a “System Cell”, ENFMOTN = (value), is created, with “value” equal to 0 for “Constraint Mode”, 1 for “Absolute”, or 2 for “Absolute, Viscous Damping”. For MSC/MD Nastran, PARAM,ENFMOTN,ABS is created for “Absolute”, while PARAM,ENFMOTN,REL is created for “Relative”.

52. For NX Nastran only, FEMAP can write out specialized Plane Strain or Plane Stress elements by setting the element formulation to “1..CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8” (Plane Strain) or “2..CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8” (Plane Stress).

53. For NEi Nastran only, the XZ Plane, YZ Plane, and the options in the Region Options section (Characteristic Length and Exact Integration Factor) on the MFLUID are not available.

54. For NEi Nastran only, setting the value to -2..Automatic(Statics) will write out PARAM,INREL,AUTO.

55. For NEi Nastran only, when the “Rigid Element Thermal Expansion” option in on, two additional PARAMs are written to Bulk Data, PARAM,RIGIDELEM2ELAS,ON and PARAM,RIGIDELEMTYPE,BAR. In conjunction with this, RIGID = LAGRAN is NOT written to the Case Control section.

56. For NX Nastran only, is a “shape-memory alloy” material.

57. For NEi Nastran only, is a “shape-memory alloy” material, specifically, Nitinol.

58. For NX Nastran only, allows you to enter Mullins effect for hyperelastic materials for SOL 601/701

59. For NX Nastran only, allows you to enter Viscoelastic effect for hyperelastic materials for SOL 601/701

60. For NX Nastran only, is a non-hyperelastic Viscoelastic material for SOL 601

61. For NX Nastran only, is a property which does not reference a hyperelastic material for Plane Strain or Plane Stress Elements. The formulation of the elements must be set to “1..CPLSTN3, CPLSTN4, CPLSTN6, CPLSTN8” (Plane Strain) or “2..CPLSTS3, CPLSTS4, CPLSTS6, CPLSTS8” (Plane Stress) in order to export this property type. The “Mean Dilatational Formulation” switch on the property may be used for nearly incompressible materials, but is ignored for SOL 601. Also, Nonstructural mass/are is ignored for SOL 601.

62. For NEi Nastran only, instructs the modal solution which eigenvalue extraction method to use. Choices are Lanczos, Subspace, and Auto.

63. For NX Nastran only, used to place pressure loads on the edge of CPLSTNi and CPLSTSi elements.

64. For NX Nastran and MSC Nastran only, used to create an acceleration load which varies along a particular axis of a specified coordinate system. Requires creation of an “Acceleration vs. Location” function (Type = 36).

65. For NX Nastran and MSC Nastran only. Selection of *Ply Failure Theory* and *Bond Failure Theory*, along with specification of limits for each is done using the *Ply/Bond Failure* tab of the *Define Material* dialog box.

66. For NX Nastran and MSC Nastran only. Created using *Ply/Bond Failure* tab for *Isotropic*, *3D Orthotropic*, and *3D Anisotropic* material types. Used only for *Solid Laminate* Elements.

67. For NX Nastran and MSC Nastran only.

68. The SUBCOM subcase combination does not support thermal loads defined in the referenced subcases. FEMAP will NOT automatically move the loads but instead warn the user that these loads need to be manually redefined as a load directly in the SUBCOM subcase.

69. For MSC Nastran only. SMETHOD = MATRIX will be written for when “Iterative Solver” is set to “1..On” in the *NASTRAN Executive and Solution Options* dialog box, while SMETHOD = ELEMENT will be written when “Iterative Solver” is set to “2..Elemental Iter”.

70. The NX Nastran solver has a predefined limit of 10,000 individual ACCEL1 entries. If more than 10,000 nodal acceleration loads are defined in a single load set, then the nodal acceleration loads will be combined, when possible, to minimize the overall number of ACCEL1 entries written to the Nastran input file. If more than 10,000 “unique” nodal acceleration loads are defined in a single load set, they will be combined into a few ACCEL1 entries as possible, then written to the Nastran file.

71. For MSC Nastran only. The BCONTACT entry is written to the Case Control section and in most cases points to a BCTABLE which contains the various “contact pairs”. For BCTABLE, the HHHB item, used for Heat Transfer, is not supported. The NLPARM entry, which must be explicitly enabled in the “Master” case or individual subcase(s), is only used when Linear contact is defined and is used for convergence purposes.

72. For NX Nastran only. MGRID is used to specify a specific node and MDOF a degree of freedom (1-6) to monitor during a direct frequency or direct transient response, plotted in the NX Nastran Analysis Monitor.

73. For NX Nastran only. SWPANGLE is angular increment in degrees at which failure indices and strength ratios are computed and output for laminates in direct frequency (SOL 108) and modal frequency (SOL 111) analysis.

7.1.4 MSC Patran Translation Notes

1. FEMAP Coordinate System 0 is not written to the MSC Patran file, but Coordinate Systems 1 and 2 are always written.

2. All coordinates are written to MSC Patran in global rectangular coordinates (CSys 0).

3. All element types are written with the appropriate shape and number of nodes to match your definition in FEMAP. The MSC Patran Element Configuration (CONFIG) is set to the value shown (Cfg=).

4. Properties are written to (and read from) the MSC Patran neutral file in the same format used by the NASPAT translator. The property values match the order used by Nastran. This format may be incompatible with interfaces from MSC Patran to other programs.
5. Constraints are written to the MSC Patran neutral file as enforced displacement loads. When reading a neutral file, FEMAP converts an enforced displacement of 0.0 to a constraint, and all others to loads.
6. Only RB2 and RBE3 elements will be imported.

7.2 Translation Table for ABAQUS, LS-DYNA, and MSC.Marc

FEMAP	ABAQUS		LS-DYNA	MSC.Marc
	Write	Read	Write	Write
Coordinate Systems				
Rectangular	---	---	*DEFINE_COORDINATE_SYSTEM	---[1]
Cylindrical	---	---	---	---[1]
Spherical	---	---	---	---[1]
Nodes				
All	*NODE,*TRANSFORM[1]	<——	*NODE	COORDINATES[1] TRANSFORM CYLIN-DRICAL
Elements				
Rod	T2D2, T2D2H, T3D2, T3D2H, DC1D2, C1D2, C1D2H[4]	<——	*ELEMENT_BEAM[1]	9
Bar	B21, B23, B21H, B23H, B31, B33, B31H, B33H[2] *RELEASE	<——	*ELEMENT_BEAM[1] *ELEMENT_BEAM_ORIENTATION[7]	52, 98[3]
Tube	PIPE21, PIPE21H, PIPE31, PIPE31H[5]	<——	*ELEMENT_BEAM[1]	14, 31[2]
Curved Tube	ELBOW31[5]	<——	---	31
Link	---	---	---	---
Beam	B21, B23, B21H, B23H, B31, B33, B31H, B33H[2] *RELEASE[12] B22, B22H, B32, B32H	<——	*ELEMENT_BEAM[1] *ELEMENT_BEAM_ORIENTATION[7]	52, 98[3]
Spring/Damper	SPRINGA, DASHPOTA, MATRIX INPUT, MATRIX ASSEMBLE[6]	<——	*ELEMENT_DISCRETE	---
DOF Spring	SPRING2, DASHPOT2[6]	<——	---	SPRINGS
Curved Beam	---	---	---	31
Gap	GAPUNI[7]	<——	---	---
Plot Only	R2D2, RAX2, RB2D2, ISL21, ISL21A, ISL31, IRS21, IRS21A[9]	R2D2, RAX2, RB2D2, RB3D2, SAX1, FAX2, SAXA1	---	---
Shear	---	---	---	68
Parabolic Shear	---	---	---	---
Membrane	M3D3, M3D4, M3D4R[8]	<——	*ELEMENT_SHELL[2] *ELEMENT_SHELL_BETA *ELEMENT_SHELL_OFFSET	18
Parabolic Membrane	M3D6, M3D8, M3D8R[8]	<—— (+ M3D8R, M3D9, M3D9R)	*ELEMENT_SHELL[2] *ELEMENT_SHELL_BETA *ELEMENT_SHELL_OFFSET	30
Bending	---	---	---	---
Parabolic Bending	---	---	---	---
Plate	S3, S4, STRI3, STRI35, S3R, S4R, S4RF, S4R5, DS4, DS3[8]	S3, S3R, S4, S4R, S4R5, STRI3, STRI35, DS4, DS3	*ELEMENT_SHELL[2] *ELEMENT_SHELL_BETA *ELEMENT_SHELL_THICKNESS *ELEMENT_SHELL_OFFSET	75, 138, 139, 140[3]
Parabolic Plate	STRI65, S8R, S8R5[8], DS6, DS8	STRI65, S8, S8R, S8R5, S9 DS6, DS8	*ELEMENT_SHELL[2] *ELEMENT_SHELL_BETA *ELEMENT_SHELL_THICKNESS *ELEMENT_SHELL_OFFSET	22
Plane Strain	CPE3, CPE4, CPE4I, CPE4R, CPS3, CPS4, CPS4I, CPS4R, DC2D3, DC2D4	CPE3, CPE4, CPE4H CPE4R, CPE4RH, CPE4I, CPE4IH, CGPE6, CPS3, CPS4, CPS4I, CPS4R, DC2D3, DC2D4	*ELEMENT_SHELL[2] *ELEMENT_SHELL_BETA *ELEMENT_SHELL_OFFSET	6, 11, 115[3], 3, 114
Parabolic Plane Strain	CPE6, CPE8, CPE8R and hybrid[8], CPS6, CPS8, CPS8R, DC2D6, DC2D8	CPE6, CPE6H, CPE8, CPE8H, CPE8RH, CPS6, CPS6M, CPS8, CPS8R, CGPE8, DC2D6, DC2D8	*ELEMENT_SHELL[2] *ELEMENT_SHELL_BETA *ELEMENT_SHELL_OFFSET	27, 54, 125[3], 32, 58, 128 26, 53, 124
Laminated Plate	STRI3, STRI35, S4, S4R, S4RF, S4R5[8,10]	<—— + (S3, S4, S4R, S4R5)	*ELEMENT_SHELL[2] *ELEMENT_SHELL_BETA	75[3],138, 139, 140
Parabolic Laminated Plate	STRI65, S8R, S8R5[8,10]	<—— (+ STRI6, S9R5)	---	22
Planar Plot Only	R2D3, R3D4, IRS3, IRS4[9]	<—— +(R3D3, F3D3, F3D4)	---	---

FEMAP	ABAQUS		LS-DYNA	MSC.Marc
	Write	Read	Write	Write
Axisymmetric	CAX3, CAX4, CAX4I, CAX4R, hybrid, DCAX3, DCAX4, CGPE6, CGPE6R, CGPE6RH, CGPE6H, CGPE6I, CGPE6IH, CGPE5, CGPE5H, SAX1[14]	<—— + (CAXA4, CGAX3, CGAX4 hybrid and reduced integration), SAX1	*ELEMENT_SHELL[2] *ELEMENT_SHELL_BETA	2, 10[3], 116, 20
Parabolic Axisymmetric	CAX6, CAX8, CAX8R and hybrid, DCAX6, DCAX8, CGPE10, CGPE10R, CGPE10H, CGPE10HR, CGPE8, CGPE8H, SAX2[14]	<—— + (CAXA8, CGAX6, CGAX8) hybrid and reduced integration), SAX2	---	28, 55[3], 66, 67, 33, 59 126, 129
Solid	C3D4, C3D6, C3D8, C3D8I, C3D8R and hybrid[3], DC3D4, DC3D6, DC3D8	C3D4, C3D6, C3D8, C3D8I, C3D8R and hybrid[3]	*ELEMENT_SOLID	7, 134[3], 117
Parabolic Solid	C3D10, C3D15, C3D20, C3D20I, C3D20R and hybrid[3], DC3D10, DC3D15, DC3D20	<—— + (C3D27R and hybrid) - (DC3D10, DC3D15, DC3D20)	---	21, 57, 127[3],35, 61, 130
Mass	MASS, ROTARYI[3,11]	<——	*ELEMENT_MASS-	MASSES
Mass Matrix	---	---	---	---
Spring/Damper to Ground	SPRINGA, DASHPOTA, MATRIX INPUT, MATRIX ASSEMBLE[6]	<——	---	---
DOF Spring to Ground	SPRING1, DASHPOT1[6]	<——	---	SPRINGS
Rigid	*KINEMATIC COUPLING[16] *MPC[17]	<——	*CONSTRAINED_NODAL_RIGID_BODY[6] *CONSTRAINED_INTERPOLATION	TYING[4]
General Matrix	MATRIX INPUT, MATRIX ASSEMBLE	---	---	---
Contact	*CONTACT PAIR	<——	*CONTACT	CONTACT TABLE
Slide Line	---	---	---	---
Weld/Fastener	---	---	---	---
Properties				
Rod	*SOLID SECTION	<——	*SECTION_BEAM	GEOMETRY
Bar	*BEAM GENERAL SECTION, *SECTION POINTS, *CENTROID, *SHEAR CENTER	<——	*SECTION_BEAM	BEAM SECT
Tube	*BEAM SECTION (PIPE)	<——	*SECTION_BEAM	GEOMETRY
Curved Tube	*BEAM SECTION (ELBOW)	<——	---	GEOMETRY
Link	---	---	---	---
Beam	*BEAM GENERAL SECTION, *SECTION POINTS, *CENTROID, *SHEAR CENTER	<—— + (*TRANSVERSE SHEAR STIFFNESS)	*SECTION_BEAM	BEAM SECT GEOMETRY
Spring	*SPRING, *DASHPOT	<——	*MAT_SPRING_ELASTIC, *MAT_DAMPER_VISCOUS	---
DOF Spring	*SPRING, *DASHPOT	<——	---	SPRING
Curved Beam	---	---	---	BEAM SECT GEOMETRY
Gap	*GAP, *FRICTION	<——	---	---
Plot Only	---	---	---	---
Shear	---	---	---	GEOMETRY
Parabolic Shear	---	---	---	GEOMETRY
Membrane	*SOLID SECTION	<——	*SECTION_SHELL	GEOMETRY
Parabolic Membrane	*SOLID SECTION	<——		GEOMETRY
Bending	---	---	---	GEOMETRY
Parabolic Bending	---	---	---	GEOMETRY
Plate	*SHELL SECTION	<——	*SECTION_SHELL	GEOMETRY
Parabolic Plate	*SHELL SECTION	<——	*SECTION_SHELL	GEOMETRY
Plane Strain	*SOLID SECTION	<——	*SECTION_SHELL	GEOMETRY
Parabolic Plane Strain	*SOLID SECTION	<——	---	GEOMETRY

FEMAP	ABAQUS		LS-DYNA	MSC.Marc
	Write	Read	Write	Write
Laminated Plate	*SHELL SECTION	<——	*PART_COMPOSITE *SECTION_SHELL (pre-version 11.1.1) *INTEGRATION_RULE (pre-version 11.1.1)	GEOMETRY COMPOS-ITE
Parabolic Laminated Plate	*SHELL SECTION	<——	---	GEOMETRY COMPOS-ITE
Axisymmetric	*SOLID SECTION	<——	*SECTION_SHELL	GEOMETRY
Parabolic Axisymmetric	*SOLID SECTION	<——	---	---
Solid	*SOLID SECTION	<——	*SECTION_SOLID	GEOMETRY
Parabolic Solid	*SOLID SECTION	<——	---	---
Mass	*MASS, *ROTARYINERTIA	<——	*ELEMENT_MASS	MASSES
Mass Matrix	---	---	---	---
Spring/Damper to Ground	SPRING, DASHPOT, MATRIX INPUT, MATRIX ASSEM-BLE[6]	<——	---	---
DOF Spring to Ground	SPRING, DASHPOT[6]	<——	---	SPRINGS
Rigid	none	none	---	none
General Matrix	MATRIX INPUT, MATRIX ASSEMBLE	<——	---	---
Slide Line	---	---	---	---
Weld/Fastener	---	---	---	---
Materials				
Isotropic	*MATERIAL, *ELASTIC (ISO), *EXPANSION, *DENSITY, *SPECIFIC HEAT, *DAMPING, *CONDUCTIVITY, *PLASTIC[15], *CREEP, *DRUCKER PRAG, *PLASTIC, *MOHR COULOMB	*MATERIAL, *ELASTIC (ISO), *EXPANSION, *DENSITY, *SPECIFIC HEAT, *DAMPING, *CONDUCTIVITY, *PLASTIC[15], *DRUCKER PRAGER, *DRUCKER PRAGERH, *MOHR COULOMB, *MOHRCOULOMBH	ELASTIC, ISOTROPIC_ELASTIC_ PLASTIC, PLASTIC_KINEMATIC, ELASTIC_PLASTIC_THERMAL	ISOTROPIC, WORK HARDENING, TEMPERATURE EFFECTS, STRAIN RATE
Orthotropic -2D	*MATERIAL, *ELASTIC (LAMINA), *EXPANSION, *DENSITY, *SPECIFIC HEAT, *DAMPING, *CONDUCTIVITY, *PLASTIC[15], *CREEP, *DRUCKER PRAG, *MOHR COULOMB	*MATERIAL, *ELASTIC (LAMINA), *EXPANSION, *DENSITY, *SPECIFIC HEAT, *DAMPING, *CONDUCTIVITY, *PLASTIC[15], *DRUCKER PRAGER, *DRUCKER PRAGERH, *MOHR COULOMB, *MOHRCOULOMBH	ORTHOTROPIC_ELASTIC, COMPOSITE_DAMAGE	ORTHOTROPIC, WORK HARDENING, TEMPERATURE EFFECTS, STRAIN RATE
Orthotropic -3D	*MATERIAL, *ELASTIC (ENGR CONST), *EXPANSION, *DENSITY, *SPECIFIC HEAT, *DAMPING, *CONDUCTIVITY, *PLASTIC[15], *CREEP, *DRUCKER PRAG *MOHR COULOMB	*MATERIAL, *ELASTIC (ENGINEERING CONSTANTS, ORTHRO), *EXPANSION, *DENSITY, *SPECIFIC HEAT, *DAMPING, *CONDUCTIVITY, *PLASTIC[15], *DRUCKER PRAGER, *DRUCKER PRAGERH *MOHR COULOMB, *MOHRCOULOMBH	ORTHOTROPIC_ELASTIC[3], COMPOSITE_DAMAGE	ORTHOTROPIC, WORK HARDENING, TEMPERATURE EFFECTS, STRAIN RATE
Anisotropic -2D	*MATERIAL, *ELASTIC(ANISO), *EXPANSION, *DENSITY, *SPECIFICHEAT, *DAMPING, *CONDUCTIVITY, *PLASTIC[15], *CREEP, *DRUCKER PRAG, *PLASTIC *MOHR COULOMB	*MATERIAL, *ELASTIC (ANISO), *EXPANSION, *DENSITY, *SPECIFIC HEAT, *DAMPING, *CONDUCTIVITY, *PLASTIC[15], *DRUCKER PRAGER, *DRUCKER PRAGERH, *MOHR COULOMB, *MOHR COULOMBH	ANISOTROPIC_ELASTIC	ANISOTROPIC, WORK HARDENING, TEMPERATURE EFFECTS, STRAIN RATE

FEMAP	ABAQUS		LS-DYNA	MSC.Marc
	Write	Read	Write	Write
Anisotropic -3D	*MATERIAL, *ELASTIC(ANISO), *EXPANSION, *DENSITY, *SPECIFIC HEAT, *DAMPING, *CONDUCTIVITY, *PLASTIC[15], *CREEP, *DRUCKER PRAG, *MOHR COULOMB	*MATERIAL, *ELASTIC (ANISO), *EXPANSION, *DENSITY, *SPECIFIC HEAT, *DAMPING, *CONDUCTIVITY, *PLASTIC[15], *DRUCKER PRAGER, *DRUCKER PRAGERH, *MOHR COULOMB, *MOHR COULOMBH	ANSIOTROPIC_ELASTIC[3]	ANISOTROPIC, WORK HARDENING, TEMPERATURE EFFECTS, STRAIN RATE
Hyperelastic	*HYPERELASTIC	<——	---	MOONEY
Other	*HYPERELASITIC, OGDEN (200) *HYPERFOAM(201)	<——	*MAT (various values between 1 to 181)[3]	OGDEN (300), FOAM (301)
Functions				
All	*AMPLITUDE	<——	*DEFINE_CURVE	
Constraints				
Nodal	*BOUNDARY	<——	*BOUNDARY_SPC_NODE	FIXED DISP, DISP CHANGE
Nodal NonZero	---	---	*BOUNDARY_PRESCRIBED_MOTION_NODE	---
Equation	*EQUATION	<——	*CONSTRAINED_LINEAR[4]	SERVO LINK
Nodal Loads				
Force and Moment	*CLOAD	<——	*LOAD_NODE_POINT	POINT LOAD
Displacement	*BOUNDARY,TYPE= DISPLACEMENT (not 0.)	<——	*BOUNDARY_PRESCRIBED_MOTION	FIXED DISP, DISP CHANGE
Velocity	*BOUNDARY, TYPE= VELOCITY	<——	*BOUNDARY_PRESCRIBED_MOTION, *BOUNDARY_INITIAL_VELOCITY_NODE	INITIAL VELO
Acceleration	*BOUNDARY, TYPE= ACCELERATION	<——	*BOUNDARY_PRESCRIBED_MOTION	FIXED ACCE
Temperature	*BOUNDARY (DOF 11), *TEMPERATURE	<——	*INITIAL_TEMPERATURE_ LOAD, *LOAD_THERMAL_CONSTANT_NODE	POINT TEMP INITIAL TEMP
Heat Generation	---	---	---	---
Heat Flux	*CFLUX	---	---	---
Nonlinear Transient	---	---	---	---
Elemental Loads				
Distributed Load	*DLOAD(PX, PY, PZ, P1, P2)	<——	LOAD_BEAM_ELEMENT	DISP LOADS (1,2,3)
Pressure	*DLOAD (P)	<——	*LOAD_SEGMENT, *LOAD_SHELL_ ELEMENT	DISP LOADS
Temperature	---	---	---	---
Heat Generation	*DFLUX(BF)	---	---	---
Heat Flux	*DFLUX(Sn)	---	---	---
Convection	*FILM	---	---	---
Radiation	*RADIATE	---	---	---
Body Loads				
Translational Acceleration	*DLOAD (GRAV)	<——	LOAD_BODY_X[5] LOAD_BODY_Y LOAD_BODY_Z	DISP LOADS (102)
Rotational Acceleration	*DLOAD (ROTA)	<——	---	---
Rotational Velocity	*DLOAD(CENT)	<——	LOAD_BODY_RX[5] LOAD_BODY_RY, LOAD_BODY_RZ, *INTIAL_VELOCITY _GENERATION	DISP LOADS (103)
Rotation Origin	---	---	On Above	ROTATION A
Default Temperature	*INITIAL CONDITIONS, TYPE= TEMPERATURE, VELOCITY	<——	---	---
Heat Transfer	*PHYSICAL CONSTANTS	---	---	---

FEMAP	ABAQUS		LS-DYNA	MSC.Marc
	Write	Read	Write	Write
Miscellaneous				
CONTACT	---	---	*CONTACT, *CONTACT_AUTOMATIC_GENERAL, *CONTACT_AUTOMATIC_SURFACE_TO _SURFACE, *CONTACT_CONSTRAINT_SURFACE_TO _SURFACE, *CONTACT_ERODING_SURFACE_TO_SURFACE, *CONTACT_FORCE_TRANSDUCER_PENALTY, *CONTACT_FORMING_SURFACE_TO_SURFACE, *CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID _BODY, *CONTACT_SINGLE_EDGE, *CONTACT_SLIDING_ONLY_PENALTY, *CONTACT_SURFACE_TO_SURFACE, *CONTACT_TIEBREAK_SURFACE_TO_SURFACE, *CONTACT_TIED_SHELL_EDGE_TO_SURFACE *CONTACT_TIED_SHELL_EDGE_TO_SURFACE _BEAM_OFFSET, *CONTACT_TIED_SURFACE_TO_SURFACE, *DEFINE_BOX, *DEFORMABLE_TO_RIGID	CONTACT, CONTACT NODE, CONTACT TABLE
	*NSET, *ELSET, *SLIDE LINE, *INTERFACE, *RIGID SURFACE, *VERTEX, *PATCH	*NSET, *ELSET, *RIGID SURFACE	*CONTROL_SOLUTION *CONTROL_TERMINATION *CONTROL_IMPLICIT_AUTO[8] *CONTROL_IMPLICIT_GENERAL[8] *CONTROL_IMPLICIT_SOLUTION, *CONTROL_IMPLICIT_SOLVER, *CONTROL_IMPLICIT_TERMINATION, *DAMPING,_GLOBAL, _PART_STIFFNESS	DEFINE,END OPTION, CONTINUE, DYNAMIC RECOVER, MODAL SHAPE, BUCKLE
	*ORIENTATION, *NORMAL	*ORIENTATION	---	CONTROL, SOLVER, AUTO INCREMENT AUTO STEP
	*STEP, *END STEP, *AMPLITUDE, *VISCO, *STEADY STATE DYNAMICS, *MODAL DYNAMIC, *DYNAMIC	*STEP, *END STEP, *AMPLITUDE	*SET,_SOLID_LIST, _SHELL_LIST, _PART_LIST, _NODE_LIST, _BEAM_LIST	PLASTICITY, ELASTICITY, LARGE DISP, UPDATE, ELASTIC
	*HEADING, *STATIC, *FREQUENCY, *BUCKLE, *EL PRINT, *NODE PRINT, *FILE FORMAT, *EL FILE, *NODE FILE	*INCLUDE	*DATABASE, _BINARY, _D3PLOT, _BINARYD3THDT, _NODOUT, _ELOUT, _HISTORY	TIE, DIST LOADS, SIZ- ING, FOLLOW FOR, PROCESSOR ELSO SETNAME, TITLE, SHELL SECT

7.2.1 ABAQUS Translation Notes

1. Nodal coordinates are always written by FEMAP in the global Cartesian coordinate system, no matter how they are defined in FEMAP. However, if output coordinate systems are selected, additional *TRANSFORM commands are written to properly orient the degrees of freedom.

2. Beam/bar offsets are supported, but all offsets at both ends of the element must be equal, and they must not have any components along the length of the beam. Nonstructural mass and tapered beams are likewise not supported. Beam/bar offsets can only be applied to ABAQUS 3-D beam elements.

3. Solid elements can use coordinate systems to orient their material axes, and mass elements use coordinate systems to orient their inertial axes. The coordinate systems that you pick are written as *ORIENTATION commands. You should not typically use spherical coordinate systems for this purpose because the ABAQUS spherical degrees of freedom are different than the FEMAP degrees of freedom.

4. Rod elements do not support torsional properties or nonstructural mass.

ABAQUS link elements C1D2 and C1D2H were replaced by the TnDn series of truss elements in ABAQUS v5.4. C1D2 and C1D2H elements will only be written if pre-version 5.4 format is checked.

5. Tube and curved tube elements do not support nonstructural mass.

6. Spring/dashpot elements are axial only - torsional springs/dampers are not supported.

FEMAP Spring/Damper elements, when the *Spring/Damper* property has *Type* set to *Other (NASTRAN CROD/CVISC)* and FEMAP DOF spring elements must be defined in one of two ways. Elements that reference a property card with a non-zero stiffness value (property written as *SPRING) will be written as ABAQUS spring elements. Elements that reference a property with zero stiffness and non-zero damping (property written as *DASHPOT) will be written as ABAQUS dashpot elements. Both spring and dashpot properties can be specified on the same property card for export, but will be exported as two elements and two properties. DOF springs can be used to connect any degrees of freedom.

FEMAP Spring/Damper elements, when the *Spring/Damper* property has *Type* set to *CBUSH*, will create a Stiffness Matrix and/or a Viscous Damping Matrix with appropriate values. The values entered for Damping and Structural Damping in the *Spring/Damper* property are both considered when creating the Viscous Damping Matrix. The various Matrix elements are only supported for Linear Static and Modal Analysis. *Spring/Damper to Ground* elements using the *CBUSH* type and will only have the top-left quarter sub-matrix written.

7. Gap friction can be specified. It is translated to GAP FRICTION ANISOTROPIC.

8. Nonstructural mass is not supported for membrane, plate or laminate elements.

9. Plot Only and Plot Planar elements are only translated if they are selected as part of a contact segment (rigid), slide line, rigid surface or rigid body.

10. The “Options” in the *Laminate Definition* portion of the dialog box are not supported. All laminate plys must be specified directly in FEMAP.

11. A mass element can be defined in one of three ways: If you define mass and inertia on the same FEMAP property then FEMAP will create synthetic properties and elements necessary to support writing the *MASS and *ROTARY INERTIA commands. The second method requires the user define either pure mass or pure inertia. Elements that are defined with a MassX term will be translated to *MASS commands. Other mass elements will be translated to *ROTARY INERTIA commands. If *MASS is written, only MassX, I11, I22, and I33 terms are supported. If *ROTARY INERTIA is written, MassX, I11, I22, I12, I13, and I23 terms are supported. Beam releases are only supported in the elemental rotational degrees of freedom. Also, DOF 5 in FEMAP corresponds to M2 in ABAQUS and DOF 6 corresponds to M3.

12. Beam releases are only supported in the elemental rotational degrees of freedom. Also, DOF 5 in FEMAP corresponds to M2 in ABAQUS and DOF 6 corresponds to M3.

13. STRI35, ISL21, ISL21A, ISL31, and SS BEAM in ABAQUS STANDARD are not available for ABAQUS 6.1. They can be written by using *File, Export, Analysis Model* command rather than the *Analysis Set Manager* (*Model, Analysis*).

14. SAX1 and SAX2 elements must be modeled in the XY plane. The r-direction is aligned with global X-direction and the z-direction corresponds to the global Y-direction. The “top” surface of the shell is defined as the positive normal direction from node 1 to 2 of the loaded element. Pressure Loads are defined on the “top” or bottom surface of the shell. For Distributed loads the direction must set to Global X, Y, or Z.

15. The Stress/Strain curve for plasticity is defined in FEMAP using function type 14..Stress vs. Plastic Strain. ABAQUS requires that the first data pair of the function be the onset of plasticity (the plastic strain value must be zero in the first pair), for correct translation you must also define the value of "Initial Yield Stress" in the Nonlinear Material dialog box. If you require a temperature dependent Stress Strain curve you must define 3 types of functions. (**Function 1**, 14.. Stress vs. Plastic Strain for each temperature; **Function 2**, 5..Function vs Temp. The Y value is the ID of the Stress vs Plastic Strain curve and the X value is the corresponding temperature at which that curve is valid; **Function 3**, Define a type 2..vs Temperature where Y is the yield stress at each temperature X. If a yield point does not exist for all temperatures then FEMAP will interpolate this function to find a yield stress.)

16. When a rigid element is defined using formulation "0..None - Ignore" and is not an interpolation elelment, FEMAP will write *KINEMATIC COUPLING to the ABAQUS input file with the Independent Node specified as the REF NODE and all dependent nodes specified with appropriate degrees of freedom. When a formulation is specified for rigid elements, FEMAP will write them out as *MPC entries with the appropriate options.

17. All "Interpolation" elements created in FEMAP are written to ABAQUS input files as *MPC entries using the TIE option.

7.2.2 LS-DYNA Translation Notes

1. Rod, tube, bar, and beam elements are all written as LS-DYNA BEAM elements, although with different element formulations. Also, FEMAP will create synthetic nodes for all beam elements that do not have third node referenced for orientation. Also, beam offsets are not currently supported for LS-DYNA.

2. Membrane, plate, laminate, plane strain, and axisymmetric elements are all written as LS-DYNA SHELL elements.

3. In general it is best to use the *Other Type* materials for all LS-DYNA materials since this enables you to specify exactly what material to use instead of FEMAP selecting a material.

4. FEMAP writes constraint equations to LS-DYNA, but all coefficients for the DOFs of a given node in the constraint equation must have the same value.

5. Body loads can only be applied to the entire structure. Also, LS-DYNA conventions are directly opposite of FEMAP, so FEMAP will automatically reverse the sign of the body loads. Also, functionally-dependent body loads are supported in version 11.2 and above.

6. If the box next to *Factor* in the *Interpolation* section of the *Define RIGID Element* dialog box is "checked", FEMAP will export a *CONSTRAINED_INTERPOLATION entry. If not, FEMAP will simply export a *CONSTRAINED_NODAL_RIGID_BODY entry.

7. Default formulation for BAR and BEAM elements is 1..Hughes-Liu, which uses a vector to define beam orientation instead of a third node. These elements use the *ELEMENT_BEAM_ORIENTATION entry.

8. All values for *CONTROL_IMPLICIT_AUTO and *CONTROL_IMPLICIT_GENERAL can be specified by clicking the *Advanced...* button in the *Solver Options* section of the *LS-DYNA* Analysis Control dialog box.

7.2.3 MSC.Marc Translation Notes

1. Nodal coordinates referencing a rectangular coordinate system for their output coordinate system will be written in global rectangular (CSys 0) and corresponding TRANSFORM is written for those not in global rectangular. Nodes referencing a cylindrical system are written in that coordinate system with a corresponding CYLINDRICAL option.

2. Type 14 element is only chosen if Hybrid Formulation is selected.

3. For many FEMAP elements, MSC.Marc has several different types for each element. To determine which element will be used for a specific formulation, see Section 6, "Element Reference".

4. Rigid elements connecting all 6 DOFs are translated as TYING, type 100. If all 6 DOFs are not connected, separate TYING cards are written for each connected DOF.

8 Analysis Program Interfaces

This topic describes the FEMAP interfaces to specific analysis software. Each section describes how you can write or read data from an analysis software program.

Analysis Software Interfaces

The following table lists the current analysis software interfaces and supported versions.

Analysis Software or Interface File Type	Section	FEMAP Interfaces	Latest Supported Version
FEMAP neutral files	Section 8.1, "FEMAP Neutral Files"	• Write FEMAP neutral files • Read FEMAP neutral files	-
ABAQUS	Section 8.2, "ABAQUS Interfaces"	• Write FEMAP model to ABAQUS • Read ABAQUS model or analysis results into FEMAP	ABAQUS 6.14-1
ANSYS	Section 8.3, "ANSYS Interfaces"	• Write FEMAP model to ANSYS • Read ANSYS model or analysis results into FEMAP	ANSYS 17.0
I-DEAS	Section 8.4, "I-DEAS Interfaces"	• Write FEMAP model to I-DEAS universal file • Read I-DEAS model into FEMAP	I-DEAS 9.0
LS-DYNA	Section 8.5, "LS-DYNA Interfaces"	• Write FEMAP model to LS-DYNA • Read LS-DYNA mesh or analysis results into FEMAP	LS-DYNA R8
MSC Marc	Section 8.6, "Marc Interfaces"	• Write FEMAP model to MSC.Marc • Read MSC.Marc mesh or analysis results into FEMAP	MSC Marc 2005
NX Nastran	Section 8.7, "Nastran Interfaces"	• Write FEMAP model to NX Nastran • Read NX Nastran model or analysis results into FEMAP	NX Nastran 10.2
Autodesk Nastran (NEi Nastran)	Section 8.7, "Nastran Interfaces"	• Write FEMAP models • Read models or results into FEMAP	Autodesk Nastran 2016
MSC Nastran	Section 8.7, "Nastran Interfaces"	• Write FEMAP model to MSC.Nastran • Read MSC.Nastran model or analysis results into FEMAP	MSC Nastran 2014
MSC Patran	Section 8.8, "Patran Interfaces"	• Write FEMAP model to MSC.Patran • Read MSC.Patran model or analysis results into FEMAP	2.5+
CAEFEM CDA/Sprint2 CFDesign SINDA/G	Section 8.9, "Vendor-Supported Interfaces"	• Write FEMAP models • Read models or results into FEMAP	interface supported by analysis program vendor
Comma-separated tables	Section 8.10, "Comma-Separated Tables"	• Write FEMAP results as comma-separated tables • Read comma-separated tables as results into FEMAP	-

Other Analysis Program Interfaces

This table lists other available FEMAP interfaces. These interfaces are available, but are not maintained. The interfaces may not support more current versions of the analysis programs.

Analysis Program	Section	FEMAP Interfaces	Latest Supported Version
ALGOR	Section E.2, "ALGOR Interfaces" in Appendix E, *FEMAP User Guide* online help	• Write FEMAP model to ALGOR • Read ALGOR model or analysis results into FEMAP	ALGOR 11
CDA/Sprint 1	Section E.7, "MSC/PAL2 (CDA/SPRINT 1) Interfaces"	• Write FEMAP model to CDA/Sprint 1 • Read CDA/Sprint 1 models or analysis results into FEMAP	-
COSMIC NASTRAN	Section E.3, "COSMIC NASTRAN and ME/NASTRAN"	• Write FEMAP model to COSMIC NASTRAN • Read COSMIC NASTRAN models or analysis results into FEMAP	-
COSMOS	Section E.4, "COSMOS Interfaces"	• Write FEMAP model to COSMOS • Read COSMOS model or analysis results into FEMAP	COSMOS 1.71
CSA/NASTRAN	Section E.5, "CSA/NASTRAN Interfaces"	• Write FEMAP model to CSA/NASTRAN • Read CSA/NASTRAN model and results into FEMAP	CSA/NASTRAN 98
GENESIS	Section E.6, "GENESIS Interfaces"	• Write FEMAP model to GENESIS • Read GENESIS model or analysis results into FEMAP	GENESIS 2.0
ME/NASTRAN	Section E.3, "COSMIC NASTRAN and ME/NASTRAN"	• Write FEMAP model to ME/NASTRAN • Read ME/NASTRAN models or analysis results into FEMAP	-
MSC/pal2	Section E.7, "MSC/PAL2 (CDA/SPRINT 1) Interfaces"	• Write FEMAP model to MSC/pal2 • Read MSC/pal2 and CDA/Sprint 1	MDA/pal2 v. 4
mTAB*STRESS	Section E.8, "mTAB*STRESS Interfaces"	• Write FEMAP model to mTAB*STRESS • Read mTAB*STRESS model or analysis results into FEMAP	mTAB*STRESS 6.1
SSS/NASTRAN	Section E.9, "SSS/NASTRAN Interfaces"	• Write FEMAP model to SSS/NASTRAN • Read SSS/NASTRAN model or analysis results into FEMAP	-
STAAD	Section E.10, "STAAD Interfaces"	• Write FEMAP model to STAAD • Read STAAD model or analysis results into FEMAP	STAAD 2.1
STARDYNE	Section E.11, "STARDYNE Interfaces"	• Write FEMAP model to STARDYNE • Read STARDYNE model or analysis results into FEMAP	STARDYNE 4.41
UAI/NASTRAN	Section E.12, "UAI/NASTRAN Interfaces"	• Write FEMAP model to UAI/NASTRAN • Read UAI/NASTRAN model and results into FEMAP	UAI/NASTRAN 20
weCan	Section E.13, "weCan Interfaces"	• Write FEMAP model to weCan • Read weCan analysis results into FEMAP	weCan 5.0

8.1 FEMAP Neutral Files

FEMAP neutral files provide a way for you to access all of the data in your FEMAP model. If you are using your own analysis programs, or other programs that FEMAP does not directly support, you can write your own interfaces to or from the neutral file formats. Neutral files are also used to transfer information from one version of FEMAP to another. The neutral file format is described in a document contained on your FEMAP CD. The remainder of this section will concentrate on how to work with neutral files in FEMAP.

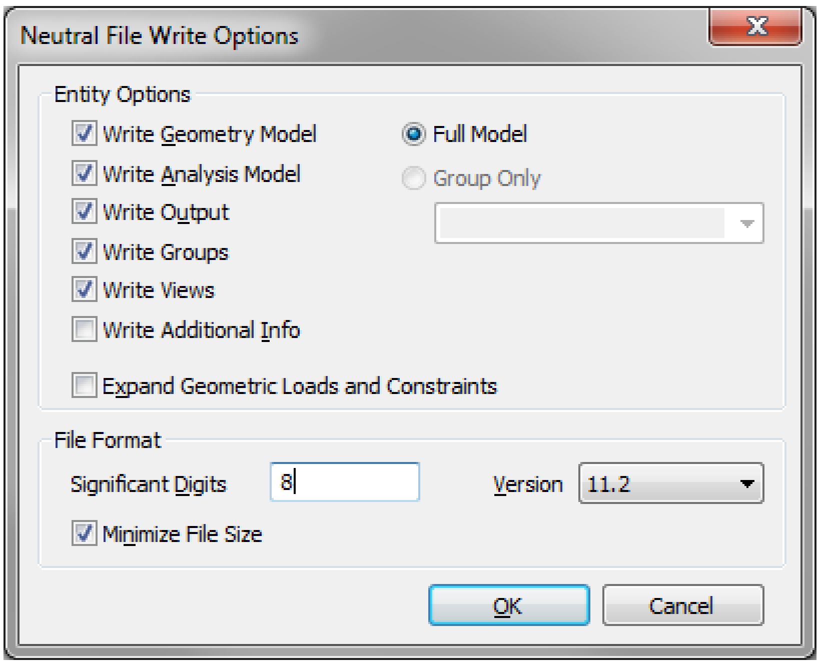
8.1.1 Writing a FEMAP Neutral File

To create a FEMAP neutral file, choose the *File, Export, FEMAP Neutral* command. You will see the standard file access dialog box, where you can choose the name of the file to create. The default extension is always ".NEU".

After you choose the file, you will see the *Neutral File Write Options* dialog box.

Entity Options

This group of options controls what data will be written to your neutral file. By default, the first five options on the left are on, so your entire model will be written. You can selectively skip parts of your model by turning one or more of these options off.



The *Write Additional Info* option will write a list of the free faces, free edges and the views which are currently on your screen. This information will not be read by FEMAP if you read the neutral file. It is only provided as information for some third-party applications that work with FEMAP.

Expand Geometric Loads and Constraints will convert all geometry-based loads into nodal and elemental loads, and all geometry-based constraints into nodal constraints. This option can be used to transfer load and constraint data which was originally geometry-based, without actually exporting the geometry with the neutral file.

Also, by default the *Full Model* option is on. This indicates that the entire model will be written.

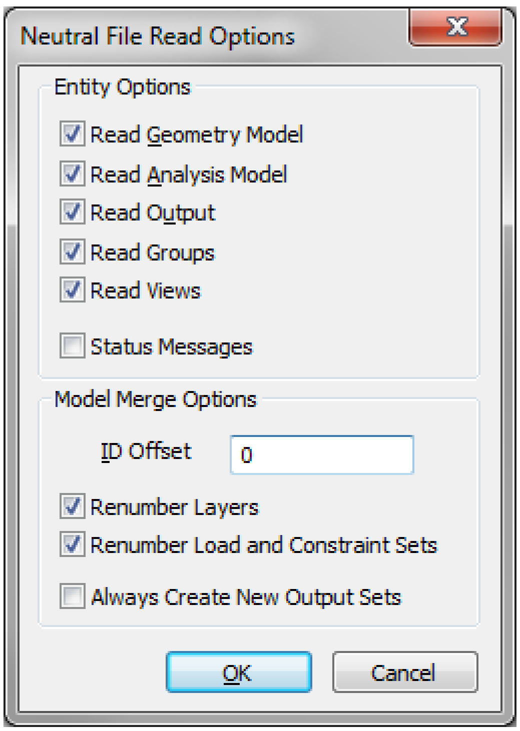
If you just want to transfer a portion of your model to the neutral file, switch to *Group Only*, and select a group from the drop-down list. Only those entities that are selected in the group (and that are enabled by the options on the left) will be translated. To work properly, you must take some care in defining the group. For example, if you translate a group that just contains certain elements, only those elements will be written to the file. The materials, properties, and nodes that those elements reference will not be written. If you try to read the neutral file into a new FEMAP model, you will get errors telling you that those entities are missing. To avoid this, you must include all of those entities in the group before you write the neutral file. One good method of doing this is to use the *Group, Node, Property,* and *Material on Element* commands to automatically group all entities referencing the chosen elements.

File Format

These options allow you to choose the format of the neutral file that you will write.

- Formatted files lose some precision, but are easily read, printed, and modified with any text editor. You can minimize the loss of precision by adjusting the number of significant digits. If you want to eliminate spaces between fields in a formatted file, turn on the *Minimize File Size* option. Leaving this option off makes the files a bit easier to look at with a text editor or print. Formatted files are always written in comma-separated, free-format. Your interfaces should never assume any particular column sizes or alignment.
- You should typically write the latest version of the neutral file to save all the data contained in the model. The only time to write older versions are if you are using an analysis program that uses an older FEMAP neutral file format, or you are transferring a model to someone who has an earlier version of FEMAP. For these cases, you should always write their specific neutral file version, because it will guarantee proper reading of the file. However, model data which did not exist in the previous version will obviously be lost in the translation.

8.1.2 Reading a FEMAP Neutral File



To read a neutral file, choose the *File, Import FEMAP Neutral* command, and choose the appropriate file from the file access dialog box. When you press *OK*, you will see the *Neutral File Read Options* dialog box.

Entity Options

The first five options are identical to similar options when writing a neutral file. They selectively allow you to skip over certain types of entities that are in the neutral file. Obviously, if you did not write those entities, they will not be read, even if these options are on. But if they are in the file, you can still bypass them by turning these options off.

Having *Status Messages* on, will NOT write the detailed inventory of what FEMAP has read in from the neutral file to the *Messages* window.

Model Merge Options

One popular use for neutral files is to merge multiple components into a single model. Since FEMAP requires unique IDs, you could have problems merging models unless you planned ahead and built them all with unique IDs. To avoid any ID conflicts, you can specify a nonzero *ID Offset* that will be added to all IDs that are read from your model. The same number is applied to all entity types.

For example, if the file you are reading has nodes and elements with IDs between 100 and 200, and you specify an *ID Offset* of 1000, the resulting FEMAP model will have nodes and elements with IDs between 1100 and 1200. By specifying a different *ID Offset* for each component model that you read, you can avoid any potential conflicts. If you are unsure of the ID ranges used by your models, the *List, Model Info* command will provide the information that you need.

Note: All of the FEMAP translators will autoscale your model when you read a file. This ensures that you can see all entities that have been read. When you read a neutral file, however, in addition to reading entities, you may be reading view information that you do not want to be autoscaled. To prevent FEMAP from changing the scale of any views, simply close all of your graphics windows prior to reading a neutral file. When the translation is complete, you can use *View, Activate* to open the views that you need.

Renumber Layers and *Renumber Load and Constraint Sets* will offset the IDs of the layers, load sets, and constraint sets so no existing layers, load sets, or constraints sets will be overwritten.

Always Create New Output Sets will read in any output sets in the neutral file as new output sets instead of overwriting output sets currently in the FEMAP model.

8.2 ABAQUS Interfaces

FEMAP provides direct interfaces to the ABAQUS file formats. Topics include:

- Section 8.2.1, "Writing an ABAQUS Model with Model, Analysis"
- Section 8.2.2, "Writing an ABAQUS Model with File, Export" (Obsolete)
- Section 8.2.3, "Performing an ABAQUS Analysis"
- Section 8.2.4, "Reading ABAQUS Models"
- Section 8.2.5, "Post-processing ABAQUS Results"

For more information on the entities that this interface translates, see:

- Section 7.2, "Translation Table for ABAQUS, LS-DYNA, and MSC.Marc"

8.2.1 Writing an ABAQUS Model with Model, Analysis

The *Model, Analysis* command opens the *Analysis Set Manager*, which enables you to write a FEMAP model to ABAQUS. The interface supports ABAQUS versions 5.4 and above. See table in "Analysis Software Interfaces" for latest supported version.

If you have an version of ABAQUS older than 5.4, use the *File, Export, Analysis Model* command instead.

8.2.1.1 Preparing the Model for Analysis

For some types of analysis (contact, nonlinear, and dynamic), you may need to set special parameters in your FEMAP model. For other types of analysis, you will follow the steps in Section 8.2.1.2, "Analysis Process Overview".

Preparing a Model for Contact Analysis

To set up a contact analysis in ABAQUS, you need to:

1. Build and mesh the FEMAP model. Apply constraints and loads.
2. Define contact in FEMAP. When you define a contact property, set the ABAQUS-specific parameters. For details, see Section 4.4, "Creating Connections and Regions"in the *FEMAP Commands Manual.*
3. Set up the analysis with the FEMAP *Analysis Set Manager*. Choose ABAQUS as the analysis program. For more details, see Section 8.2.1.2, "Analysis Process Overview".
4. On the *ABAQUS STEP Options* dialog box, set the options for contact.

Defining Contact Pair Parameters

For each contact pair, you can use the ABAQUS APPROACH and/or SLIDE DISTANCE parameters. These parameters can also be modified between steps. To use APPROACH and SLIDE DISTANCE, you must select them in two places: first, when you define the contact property, and second, when you define each individual time step. The procedure is:

1. Define the contact property: pick *Model, Contact, Contact Property.*
2. On the *Define Property* dialog box, set parameters for the property.
3. Pick the ABAQUS button.
4. On the *ABAQUS Advanced Contact Property Options* dialog box, pick the *Max Slide Distance* and/or *Approach* options. These options must be selected here if you want to use them when you define the time steps.
5. Set up the ABAQUS analysis: pick *Model, Analysis*. Choose ABAQUS as the analysis program.
6. On the *ABAQUS Step Options* dialog box, pick the *Approach* and/or *Slide Distance* options. You can change these settings in different time steps.

Preparing a Model for Nonlinear Analysis

To set up a nonlinear analysis in ABAQUS, you need to:

1. Build and mesh the FEMAP model. Apply constraints.
2. Define nonlinear load sets in FEMAP. Pick *Model, Load, Nonlinear* to set the nonlinear load set options. For

more details, see Section 4.3.5.1, "Model, Load, Nonlinear Analysis...".

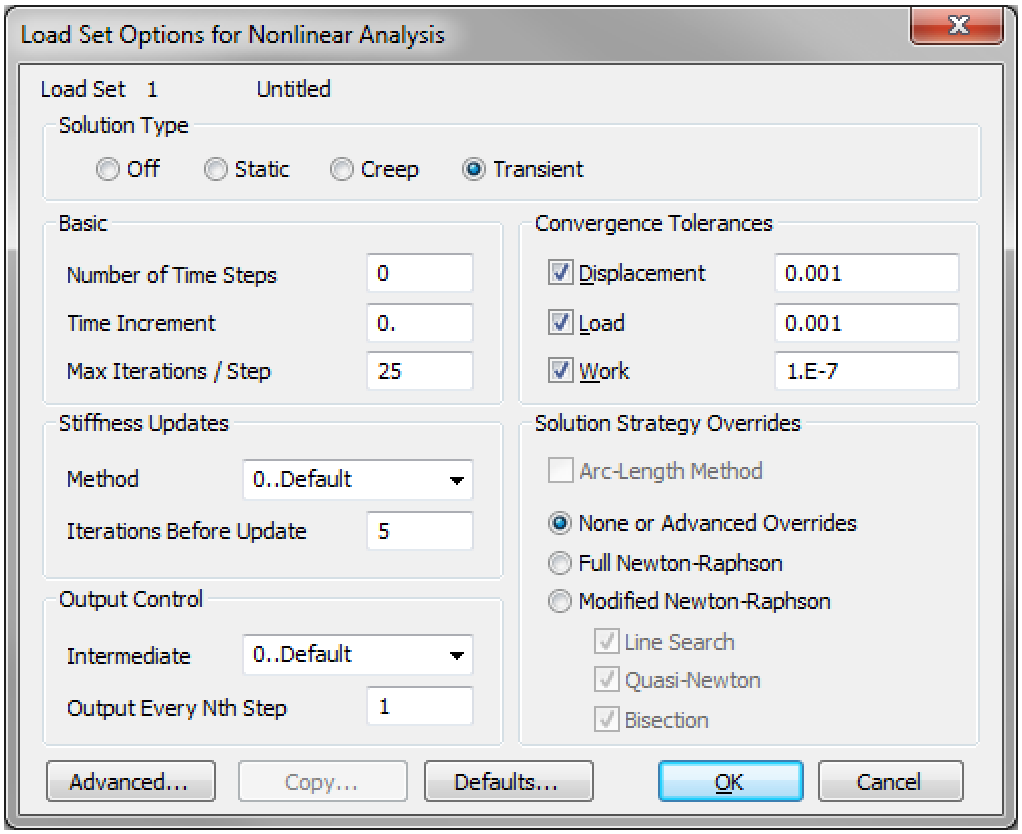
3. Set up the analysis with the FEMAP *Analysis Set Manager*. Choose ABAQUS as the analysis program. For more details, see Section 8.2.1.2, "Analysis Process Overview".

4. On the *ABAQUS STEP Options* dialog box, set the options for time steps.

Using Tolerance Control

You can use tolerance control for some nonlinear analysis types. For *HEAT TRANSFER, you can define DELTMX; for *DYNAMIC analysis, you can define HAFTOL.

To define these parameters, define a load set containing nonlinear loads:



1. Pick *Model, Load, Nonlinear* to display the *Load Set Options for Nonlinear Analysis* dialog box.

2. Pick *Transient* for the *Solution Type*.

3. Enter values for *Convergence Tolerances*.

4. Enter a value for the *Max Iterations/Step (INC=)*, if desired.

For more information, see Section 4.3.5.1, "Model, Load, Nonlinear Analysis..." in *FEMAP Commands.*

Preparing a Model for Dynamic Analysis

To set up a dynamic analysis in ABAQUS:

1. Build and mesh the FEMAP model. Apply constraints.

2. Define dynamic load sets in FEMAP. Pick *Model, Load, Dynamic* to set the dynamic load set options. For more details, see Section 4.3.5.2, "Model, Load, Dynamic Analysis..." in *FEMAP Commands.*

3. Set up the analysis with the FEMAP *Analysis Set Manager*. Choose ABAQUS as the analysis program. For more details, see Section 8.2.1.2, "Analysis Process Overview".

4. On the *ABAQUS STEP Options* dialog box, set the options for dynamic options.

If you plan to perform a transient or frequency response analysis, it is important to define the dynamic analysis options for the load set. These options control whether a direct or modal type of analysis is performed. If no solution method is chosen for the selected load case, or the solution method does not agree with the selected type (i.e.

solution method is a frequency type and you chose transient), the *STEP will not be written and FEMAP will generate an error message.

ABAQUS also requires that a *FREQUENCY analysis be performed on a previous step when choosing modal superposition. If a modal solution method is requested on the load case chosen for the *STEP and modal analysis has not been selected on a previous *STEP, FEMAP will again generate an error message and the *STEP will not be written.

Specifying Frequencies

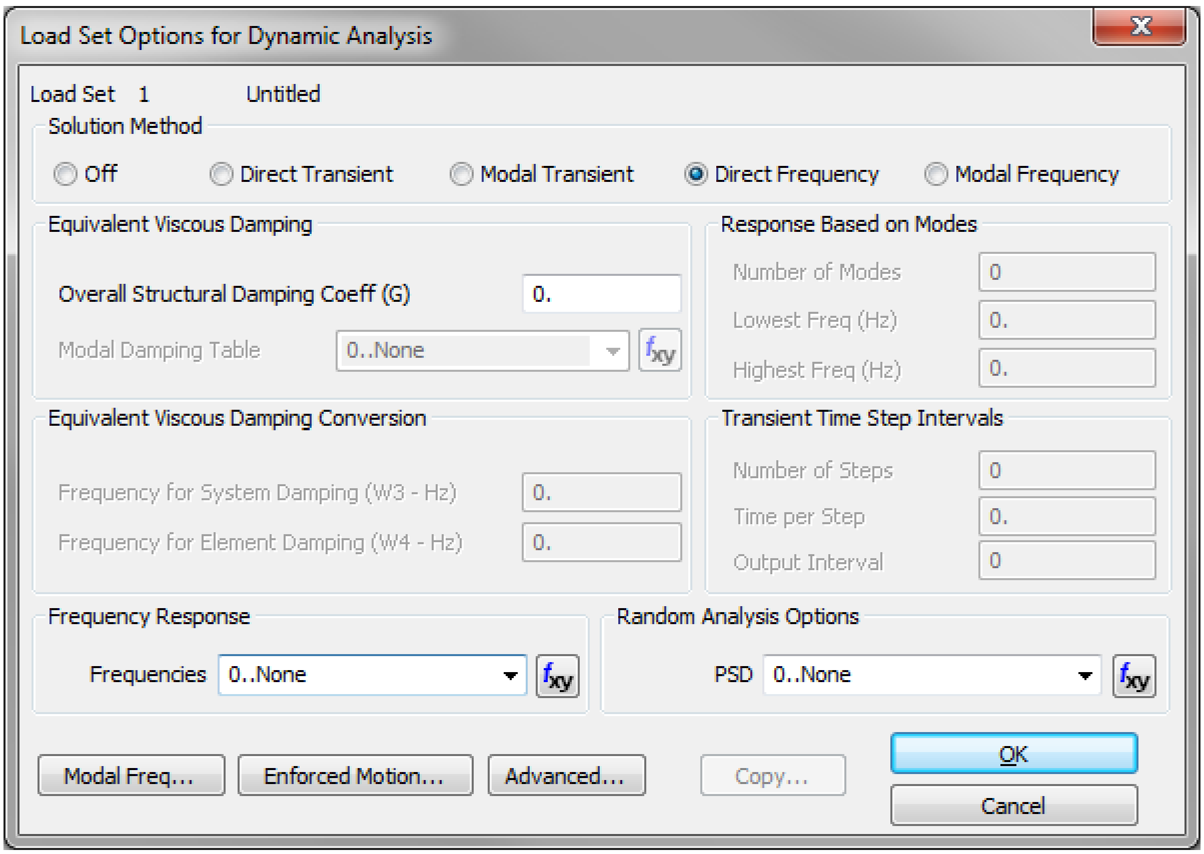
When you create the load set for the dynamic analysis in ABAQUS, you can specify a function containing frequencies of interest.

To do this:

1. Pick *Model, Function* and create a *Vs. Frequency* function.

2. Pick *Model, Load, Dynamic* to display the *Load Set Options for Dynamic Analysis* dialog box.

3. From the *Frequencies* pull-down menu, select the function. FEMAP will write the values in this table as single frequency values to be analyzed in ABAQUS.



For more information, see Section 4.3.5.2, "Model, Load, Dynamic Analysis..." in *FEMAP Commands*.

8.2.1.2 Analysis Process Overview

The *Analysis Set Manager* lets you create an analysis set, which is an input file for ABAQUS. To define an analysis set:

1. Pick *Model, Analyze* to open the *Analysis Set Manager* dialog box.

2. Pick the first item on the list, then pick *New*. (You can also double-click on the item).

3. On the *Analysis Set* dialog box, choose *16..ABAQUS* as the *Analysis Program*. Select the *Analysis Type: 1..Static, 2..Normal Modes/Eigenvalue, 3..Transient Dynamic/Time History, 4..Frequency/Harmonic Response, 7..Buckling, 9..Explicit Transient Dynamics (ABAQUS EXPLICIT), 20..Steady-State Heat Transfer,* or 21*..Transient Heat Transfer*. These options determine which element types (structural or heat transfer) will be written, and set the defaults for the other ABAQUS interface dialog boxes.

You can also choose to use a *Linked Solver* or *VisQ*. See Section 2.6.2.6, "Solvers" or Section 4.10.2.1, "Run Analysis Using Linked Solver / VisQ / Local Settings" in the FEMAP Commands Manual for more information. If no version is defined, defaults to version 10.

4. Pick *Next* to open the *ABAQUS Model Options* dialog box. (Alternatively, you can pick *OK* to close the *Analysis Set* dialog box. From the analysis set list, you can then double-click on an option to bring up the dialog box where the option is defined.) Select the version number and set options.

5. Pick *Next* to open the *Master Requests and Conditions* dialog box. Enter a title for the *Master Requests and Conditions*, which correspond to the first ABAQUS *STEP.

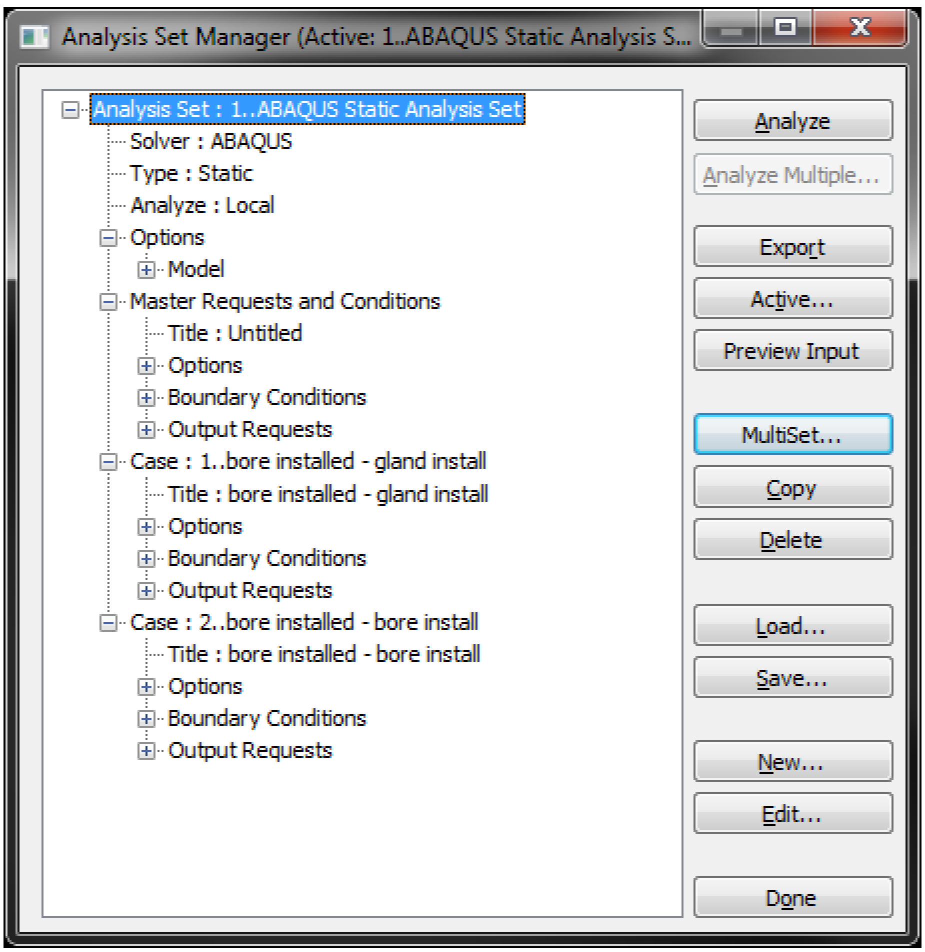
6. Pick *Next* to open the *ABAQUS Step Options* dialog box, where you can define options (such as analysis type) for the first step.

7. Pick *Next* to open the *Boundary Conditions* dialog box. Select the constraints, loads, and initial conditions that apply to the first step.

8. Pick *Next* to open the *Output Requests* dialog box. Select the output types for the first step.

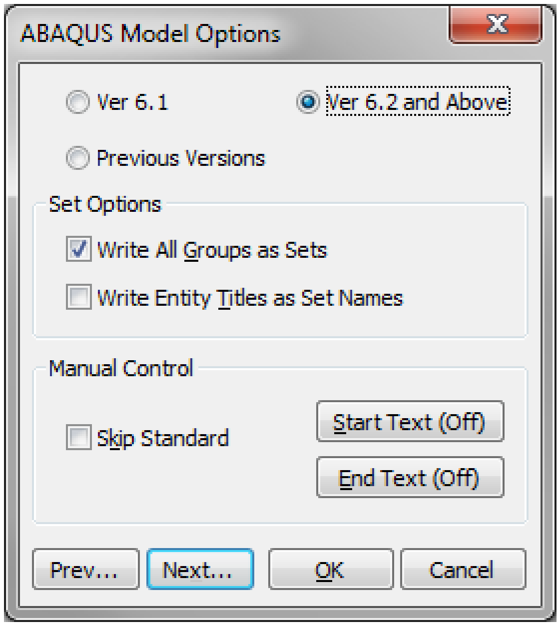
9. After the first step is created, you can create a case for each remaining step. Each case includes a title, ABAQUS options, boundary conditions, and output requests. In the *Analysis Set Manager*, double-click on *No Cases Defined*. Pick *Next* to work through the dialog boxes to create the case.

The graphic shows how an analysis set would be structured for an ABAQUS analysis with two steps.



ABAQUS Model Options

The *ABAQUS Model Options* dialog box lets you specify ABAQUS version number and set options.



Versions

Pick the version option:

- *Version 6.2 and Above*
- *Version 6.1*
- *Previous Versions*: This option supports ABAQUS versions 5.4 -6.0x. For pre-5.4 versions of ABAQUS, use the ABAQUS interface available through the *File, Export, Analysis Models* command.

Set Options

Set Options include:

- *Titles as Set Names*: If checked, this option writes ABAQUS sets to the input file using your titles for these FEMAP entities: coordinate systems, properties, materials, functions and groups. If not checked, FEMAP will automatically assign a name to these entities.
- *Write All Groups as Sets*: If checked, this option writes additional *NSET and *ELSET commands to the input file. These sets include the nodes and elements that you have selected into every group in your model; they can be very useful if you want to edit the resulting file later. Using these sets can also compact the data specified for the *SURFACE DEFINITION options.

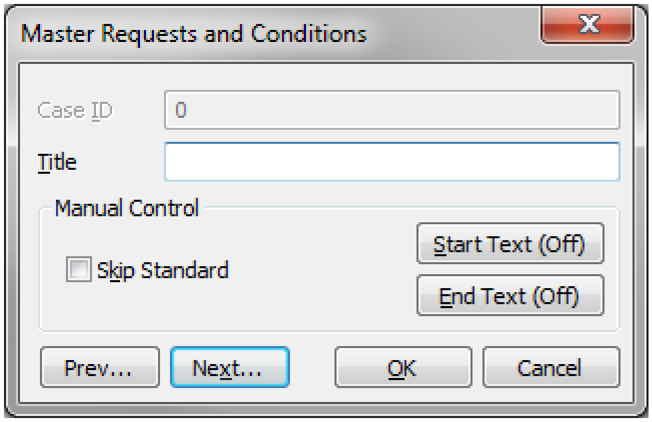
Manual Control

Manual Control options include:

- *Skip Standard*: If this switch is on, the interface does not write the standard model section in the ABAQUS input file. If *Start* and *End Text* have been defined, they will still be written to the input file.
- *Start Text*: Pick this option to add text to the beginning of the model section of the input file.
- *End Text*: Pick this option to add text to the end of the model section of the input file.

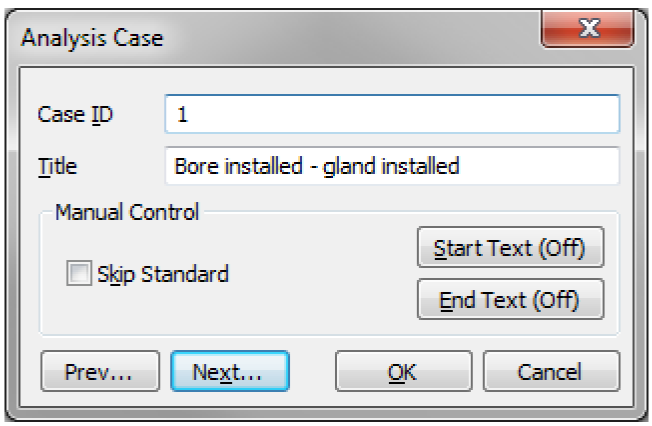
8.2.1.3 ABAQUS Master Requests and Conditions, Analysis Case

Master Requests and Conditions defines the case title for the first ABAQUS *STEP. The data you enter on the next dialog box, *ABAQUS Step Options*, further defines this step.



For each different loading condition that corresponds to an ABAQUS *STEP, you must define a new analysis case in the *Analysis Set Manager*. You must also define step options, boundary conditions, and output for each case.

The *Analysis Case* dialog box contains the same fields as the *Master Requests and Conditions* dialog box. These fields include:



Title

Enter a title for the step.

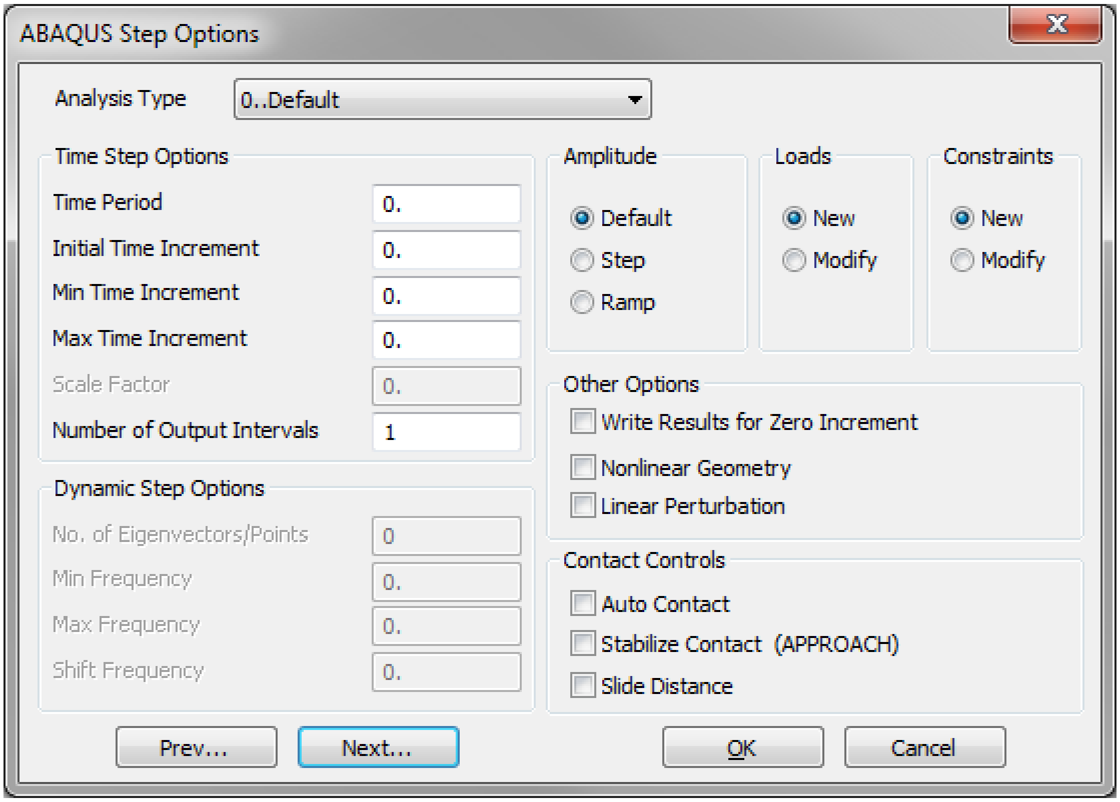
Manual Control

The *Manual Control* options include:

- *Skip Standard*: If this switch is on, the interface does not write the standard history section in the ABAQUS input file. If *Start* and *End Text* have been defined, they will still be written to the input file.
- *Start Text*: Pick this option to add text to the beginning of the history section of the input file.
- *End Text*: Pick this option to add text to the end of the history section of the input file.

8.2.1.4 *ABAQUS Step Options*

The *ABAQUS Step Options* dialog box lets you define options for each case, which corresponds on an ABAQUS step. It includes the following sections:



Analysis Type

For each case, you can specify a different analysis type, as long as the new selection is consistent with the initial element type. For example, you can switch between the various structural standard options, or between steady-state and transient heat transfer, but not between structural and heat transfer types, or between STANDARD and EXPLICIT options).

The default analysis type is defined on the *Master Requests and Conditions* dialog box.

Time Step Options

These options are translated directly to the second card of the *STATIC, *DYNAMIC, *MODAL DYNAMIC, and *HEAT TRANSFER (both steady state and transient) procedures. They control the way that ABAQUS will increment through the time step.

For most analyses, you need to:

- define the *Time Period* as the total time for the analysis
- define the *Initial Time Increment* for the analysis
- define the *Min. Time Increment* that can be used in automatic cutback
- define the *Max Time Increment*.

When translating for ABAQUS EXPLICIT, you should typically only define the *Time Period* and *Scale Factor*. You can specify a *Max Time Increment*, but this may cause the run to fail in many cases.

Note: For some analyses, you can also use tolerance control. For the detailed procedure, see Section 8.2.1.2, "Analysis Process Overview".

Dynamic Step Options

This information is written to the second card of the *FREQUENCY, *BUCKLE, and *STEADY STATE DYNAMICS options.

- For the *FREQUENCY procedure, ABAQUS will extract frequencies until it reaches either the number you specify or the *Max Frequency* limit.
- For a normal modes analysis, specify the *Max Frequency* and the *No. of Eigenvalues/Points*. You can also specify a *Shift Frequency* if you do not want to recover frequencies near 0 Hz.
- For frequency/harmonic response analyses, you can specify the number of points to be used in the analysis, as well as the minimum and maximum frequencies. These values are translated directly to the second card of the *STEADY STATE DYNAMICS option.

For additional frequencies, you can use a function when defining the load set options for dynamic analysis. For the detailed procedure, see Section 8.2.1.1, "Preparing the Model for Analysis".

Amplitude

If the loads do not have an associated time history function, the *Amplitude* options determine how these loads will be applied. These options are added to the *STEP command, and include:

- *Default*: The software bases the default step type on the analysis type.
- *Step*: This option applies full loading at the beginning of the step.
- *Ramp*: This option starts amplitude at zero, and increases the magnitude of the loads throughout the step.

Loads

Load options include:

- *New*: Check this option if you are defining a new, independent load case, or if this is the first step.
- *Modify*: Check this option if you need to apply additional loads to those already defined for a previous step, as in a perturbation analysis.

Constraints

Constraint options include:

- *New*: Check this option to use a new constraint set for the step. This set is specified on the *Boundary Conditions* dialog box.
- *Modify*: Check this option to modify the current constraint set by adding the constraint set specified in the *Boundary Conditions* dialog box. Use this option if you have a current constraint set that you want to use as a kinematic constraint set.

For modal superposition analysis types such as *MODAL DYNAMIC, and *STEADY STATE DYNAMICS (not Direct), this option is ignored since constraints cannot be modified or created in a modal superposition method. Furthermore, any constraints defined in a newly chosen constraint set for this *STEP will be ignored.

Note: The settings for the *Constraints* option will be used for all *BOUNDARY conditions in the ABAQUS STEP, including certain FEMAP nodal loads such as enforced displacements, velocities, and accelerations.

Other Options

- *Write Results for Zero Incremen*t: Check this option to write an initial condition step to the output. This can be useful when performing multi-set animation.
- *Nonlinear Geometry*: Check this option to add the NLGEOM option to the *STEP command. This will account for geometric non-linearity during this and subsequent steps, and is only relevant for stress analysis. This option is required for models which use hyperelastic materials. FEMAP will recognize that hyperelastic materials exist in the model, then check this option. If this option is checked when you enter FEMAP, deselecting it will almost always cause your analysis to fail.
- *Linear Perturbation*: Check this option to add the PERTURBATION option to *STEP. This indicates that the current step represents a change from the previous step (as opposed to an independent load step).

Contact Controls

These options write the *CONTACT CONTROLS card to ABAQUS with the following parameters:

- *Auto Contact*: This parameter writes the AUTO TOLERANCE option.
- *Stabilize Contact (APPROACH)*: This parameter activates automatic viscous damping for a contact pair.

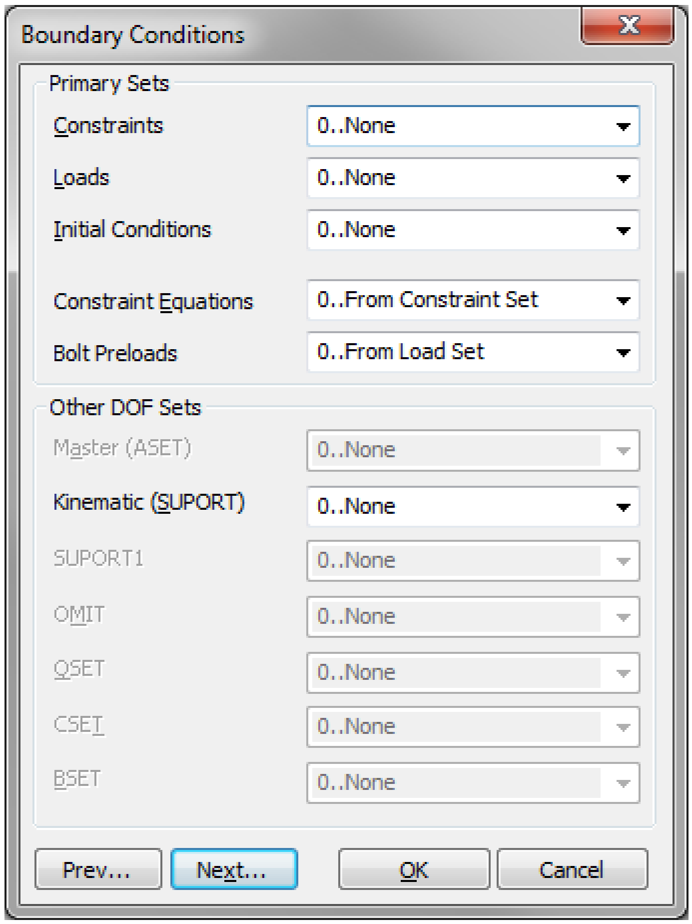
- *Slide Distance*: The SLIDE DISTANCE parameter controls the maximum slide distance for a contact pair.

The APPROACH and SLIDE DISTANCE parameters may be used for each contact pair defined in the model, and can be modified between steps. These options must be used in conjunction with the corresponding *Approach* and *Slide Distance* options when you define the contact property.

For the detailed procedure, see Section 8.2.1.1, "Preparing the Model for Analysis".

8.2.1.5 ABAQUS Boundary Conditions

The *Boundary Conditions* dialog box lets you select the loads and constraints to apply to your analysis. You can apply boundary conditions to define the first load case (in *Master Requests and Conditions*), as well as subsequent cases.



Primary Sets

Depending on your analysis type, you can select constraints and loads.

- *Constraints*: pick a constraint set for your model.
- *Loads*: pick a load set for your model
- *Initial Conditions*: pick a load set to use for initial conditions. This load set also can be used to define the frequencies for calculation of Rayleigh damping in direct transient (EXPLICIT or STANDARD) analysis. FEMAP currently supports both temperature and velocity initial conditions.

  Temperatures from the selected set will be applied to your nodes as "*INITIAL CONDITIONS, TYPE = TEMPERATURE. If the load set contains a body load default temperature, it will be applied to all the nodes of the model. If any nodal temperatures exist, they will then be applied to the appropriate nodes to redefine their initial temperatures. This will produce a warning in your ABAQUS run, but ABAQUS will utilize the appropriate temperatures. If elemental temperature loads exist in the selected load set, the software skips them and generates an error message.

You can specify velocity initial conditions in a similar manner. Any nodal velocity loads contained in the selected load set will be written as "*INITIAL CONDITIONS, TYPE=VELOCITY". Only temperatures and velocities will be utilized from the selected load set. Since load sets can be selected for each *STEP, and all loads in these sets are exported, you should create a load set containing just initial temperatures and velocities so these conditions are not exported in a *STEP along with other loads. The body load default temperature is an easy method to assign an initial temperature to all nodes in the model, which can then be redefined for any single nodes utilizing nodal temperature loads.

For direct transient analysis, FEMAP will calculate the Rayleigh damping values for each material based upon the material entry, and the frequencies input for System (W3) and Element (W4) Damping under *Model, Load, Dynamic Analysis*. The type of analysis must be selected as *Direct Transient* for FEMAP to properly convert the values. Alpha damping is computed from the product of W4 and the damping material value. Beta damping is simply the damping material value divided by W3.

- *Constraint Equations*: pick a constraint set to define constraint equations. If you choose *From Constraint Set*, FEMAP will look for constraint equations in the same set as your nodal constraints. This is a convenient way to manage most models.

For heat transfer analyses, you will notice that constraint sets are not used. Rather, loads and constraints are both selected from a load set. FEMAP translates nodal temperatures, in the same set as the other thermal loads, as thermal constraints (boundary conditions).

Other DOF Sets

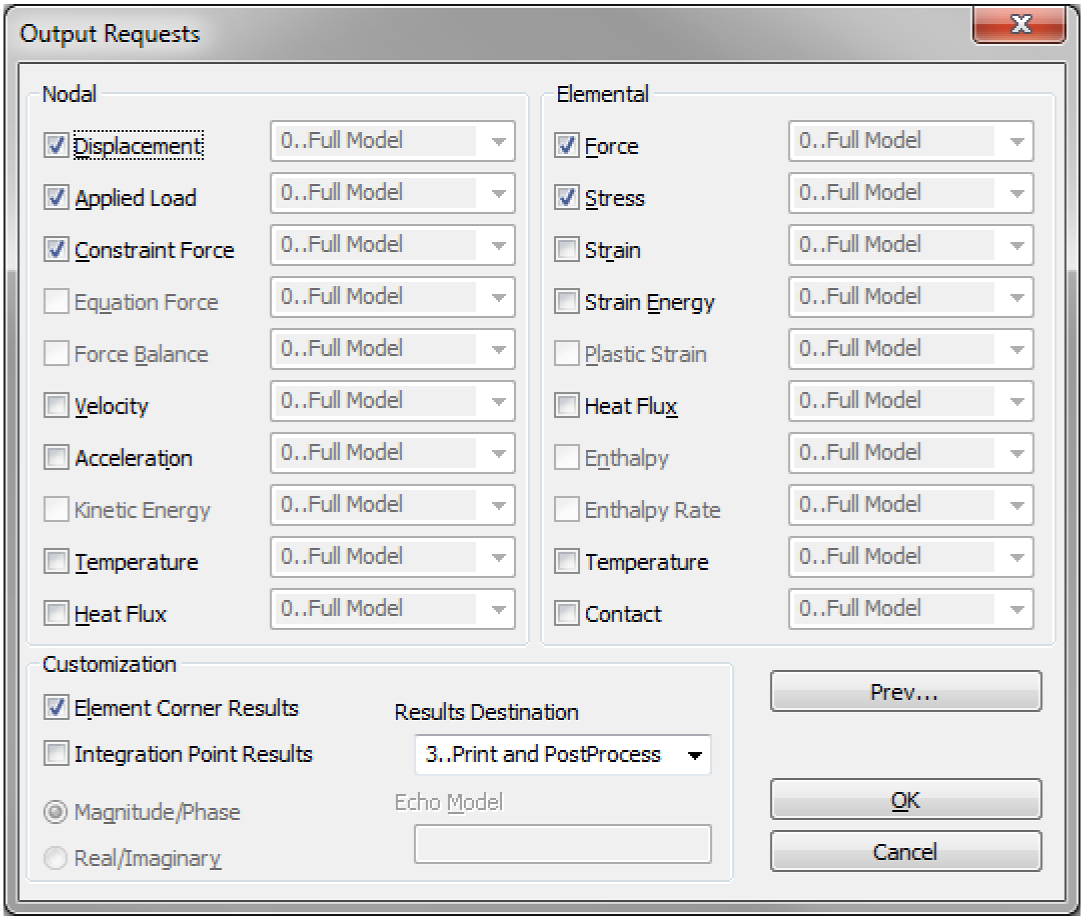
You can select constraint sets to use as various types of DOF sets.

The *Kinematic Constraint Set* option allows you to choose a constraint set that will be written prior to your first analysis *STEP. The interface combines nodal constraints from the selected set with any permanent constraints on your FEMAP nodes, and writes both. FEMAP will not write the permanent constraints unless you select a set with this option.

In addition to nodal constraints, the kinematic constraint set can also contain constraint equations which will be applied with *EQUATION commands. Any constraint equations that are in constraint sets that you select during Step Definition will be ignored. They must be specified via this option.

8.2.1.6 ABAQUS Output Requests

The *Output Requests* dialog box lets you identify the types of output that you want from the analysis. The type of output available depends on the analysis program and analysis type



You can define output requests for the master output request and subsequent cases.

Using *Results Destination*, you can choose to have results sent to “1..Print Only” (*.dat file and ABAQUS default *.odb file), “2..PostProcess Only” (*.fil or *.fin file and ABAQUS default *.odb file), “3..Print and PostProcess” (*dat file, *.fil or *.fin file, and ABAQUS default *.odb file), “7..Output Database” (User requested *.odb file), or “8..Print and Output Database” (*.dat file and user requested *.odb file). The ABAQUS default *.odb file contains output automatically selected by the solver, therefore you have no control over results written to this file.

For frequency analyses, you can recover output in either magnitude/phase or real/imaginary format.

8.2.2 Writing an ABAQUS Model with File, Export

Note: This method is obsolete and has been removed from the default configuration of FEMAP. If you need to use it for any reason, you must use the *File, Preferences* command, click the *Interfaces* tab, and check the option for “Enable Old Analysis Interfaces”. The use of this method for translation is NOT recommended. Use the *Model, Analysis* method instead.

The *File, Export, Analysis Model* command allows you to write your FEMAP model into a file that can be analyzed by ABAQUS. The interface available from this command supports ABAQUS versions 5.4-6.0x, as well as pre-5.4 versions. This interface has not been maintained for ABAQUS version 6.1. For newer versions of ABAQUS, use the *Model, Analysis* command instead.

8.2.2.1 Starting to Export

There are eight options available for ABAQUS translation: (1) *Static*, (2) *Modal Analysis*, (3) *Transient Dynamic/Time History*, (4) *Frequency/Harmonic Response*, (5) *Buckling*, (6) *Explicit Transient Dynamics* (ABAQUS EXPLICIT) (7) *Steady-State Heat Transfer*, and (8) *Transient Heat Transfer*. These options determine which element types (structural or heat transfer) will be written, and set the defaults for the dialog boxes to come. Later, you will be able to change your selection if you picked the wrong one, as long as the new selection is consistent with the initial element type (i.e. you can switch between the various structural standard options, or between steady state and transient heat transfer, but not between structural and heat Transfer types, or between STANDARD and EXPLICIT options).

Once you make your selection, you will see the standard file access dialog box, where you can specify the name of the file to be created. The default file name extension for this file is ".INP", but you can choose any name. After choosing a file name, you will see the *ABAQUS Model Write* dialog box.

This dialog box allows you to specify model definition information:

Title

Here you can specify a one-line title that will be written as an ABAQUS *HEADER command. It will appear as the page header in your print files.

Kinematic Constraint

This option allows you to choose a constraint set that will be written prior to your first analysis *STEP. Nodal constraints from the set that you select will be combined with any permanent constraints on your FEMAP nodes, and both will be written. The permanent constraints will not be written unless you select a set with this option. During the step definition phase, you can again specify constraints to be applied to the model.

In addition to nodal constraints, the kinematic constraint set can also contain constraint equations which will be applied with *EQUATION commands. Any constraint equations that are in constraint sets that you select during step definition will be ignored. They must be specified via this option.

Initial Conditions

This option allows you to select initial conditions for your model. It also can be used to define the frequencies for calculation of Rayleigh damping in direct transient (EXPLICIT or STANDARD) analysis. FEMAP currently supports both temperature and velocity initial conditions.

Temperatures from the set you select will be applied to your nodes as "*INITIAL CONDITIONS, TYPE = TEMPERATURE". If the load set contains a body load default temperature, it will be applied to all the nodes of the model. If any nodal temperatures exist, they will then be applied to the appropriate nodes, redefining their initial temperatures. This will produce a warning in your ABAQUS run, but ABAQUS will utilize the appropriate temperatures. If elemental temperature loads exist in the selected load set, they will be skipped and an error message will be printed.

Velocity initial conditions can also be specified in a similar manner. Any nodal velocity loads contained in the selected load set will be written as "*INITIAL CONDITIONS, TYPE=VELOCITY". Only temperatures and velocities will be utilized from the selected load set. Since load sets can be selected for each *STEP, and all loads in these sets are exported, you should create a load set containing just initial temperatures and velocities so these conditions are not exported in a *STEP along with other loads. The body load default temperature is an easy method for assigning an initial temperature to all nodes in the model, which can then be redefined for any single nodes utilizing nodal temperature loads.

For direct transient analysis, FEMAP will calculate the Rayleigh damping values for each material based upon the material entry, and the frequencies input for System (W3) and Element (W4) Damping under *Model, Load, Dynamic Analysis*. The direct transient analysis type must be selected in order for FEMAP to properly convert the values. Alpha damping is computed from the product of W4 and the damping material value. Beta damping is simply the damping material value divided by W3.

Overrides/Group Contact

This button allows you to access a much larger *ABAQUS Model Write* dialog box. This dialog box allows you to override your current element formulations (hybrid, 5DOF plates, etc.), as well as form contact entities via groups. All options that are available in the larger dialog box are also available by simply changing the element formulation, or by using contact pairs and segments. The group contact has been made obsolete by the contact segment/pair support. These options remain here to support old models which used this capability.

For more information, see Section 8.2.2.3, "Overrides/Group Contact".

Titles as Set Names

When this option is checked, FEMAP will use the titles given to certain entities in FEMAP when writing ABAQUS sets to the input file. This functionality is supported for coordinate systems, properties, materials, functions and groups.

Write All Groups As Sets

This option, if checked, will simply write additional *NSET and *ELSET commands to the file which represent the nodes and elements that you have selected into every group in your model. These sets are not used except when a group is selected when defining a rigid surface or contact pair. They can be very convenient if you want to edit the resulting file later, and will also compact the data on the *SURFACE DEFINITION options by using group sets.

Rigid Surfaces...

...lets you select a FEMAP group that will be translated as a rigid surface, and the associated slave surface or contact node set.

For information on how to define the group, see "Defining a Rigid Surface".

Defining a Rigid Surface

Although FEMAP does not have the equivalent of an ABAQUS rigid surface, the translator can use a FEMAP group to create one. First, however, you must properly construct your model and the group that you will use. The required contents of the group is different than the old contact modeling.

To successfully create a rigid surface, a FEMAP group must contain the following entities:

- Either one or more lines or arcs.
- One reference node.
- One or more FEMAP contact segments.
- Optionally, a coordinate system for 3-D contact.
- Optionally, a single contact property.

Lines and Arcs

When you select lines and/or arcs, they must form a continuous path. You must select the lines such that as you would walk along the lines, the contact is on your left. This can be accomplished by using *View Options, Tools and View Style, Curve and Surface Accuracy* to show the direction of the lines. The first line selected must be the start of the rigid surface, and it must be in the proper direction. FEMAP will automatically form the remaining lines in the appropriate direction.

The lines and arcs will be converted to SEGMENTS (if there are plane strain or axisymmetric elements in the model) or a CYLINDER rigid surface. Splines may not be chosen, and if you choose arcs, they must always have an included angle less than 180 degrees. If you need larger arcs, break the arc into pieces before you export.

Reference Node

You must also select a reference node for the rigid surface and one or more contact segments. The node can (and usually should) be constrained in your model. The motion, or lack thereof, of this reference node will determine the motion of the rigid body. The contact segments will be placed as the slave surfaces to the master rigid surface.

Coordinate System

A coordinate system, an optional input, is used to orient 3-D contact in space. If planar or axisymmetric elements are present, this coordinate system is unnecessary and will be ignored. If these elements are not present, thereby denoting a 3-D contact problem, FEMAP will use the coordinate system to determine the direction in which the 2-D cross-section will be extruded to form an infinite 3-D rigid surface. The X axis of the coordinate system should typically be along the length of contact, with the Y axis denoting the interface normal. The negative Z axis of the coordinate system is used to extrude the cross section. If no coordinate system is specified, FEMAP will use the Z axis as the default for the extrusion axis.

Contact Property

A contact property can also be included in the group to define the specific relationship between the rigid surface and the contact segment. This property information is exported in an identical manner as that for a contact pair.

For more information, see Section 4.2.4.4, "Other Element Properties" in *FEMAP Commands*.

8.2.2.2 Defining Analysis Steps

After you select the model translation options, you will see the *ABAQUS STEP Definition* dialog box:

It allows you to choose the various options that are available for controlling the ABAQUS analysis steps.

Load Selection

With these options, you can select the constraints and loads that will be used for the current step. In most cases, only one step will be written at a time. However, if you need to do an analysis of many independent load cases, the *All Loads as Individual Steps* option can be convenient. If it is selected, every load set in your model will be written to a corresponding step. All other options in each step will be identical, including the single constraint set you select. A selection of a constraint set will be ignored for modal procedures such as *MODAL DYNAMIC and *STEADY STATE DYNAMICS (modal), which are linear perturbation steps and do not allow you to modify the boundary conditions.

Note: All loads supported by the ABAQUS translator will be written for the load case selected. No checking is performed to determine if the load is appropriate for the analysis type chosen. Therefore, when developing load sets, be careful to place only appropriate loads for your analysis in that load set.

Applying Loads

The *Amplitude* options determine how loads will be applied for loads which do not have an associated *vs. Time* function defining their time histories. These options are added to the *STEP command. *Step* amplitude applies full loading at the beginning of the step. *Ramp* amplitude starts at zero, and increases the magnitude of the loads throughout the step. *Default* chooses whichever type is default for the type of analysis you are performing.

Similarly, loads can either be applied as a new load case, or to modify loads that are previously applied. If you are defining a new, independent load case, or if this is the first step, choose *New*. If you are trying to apply additional loads to those already defined for a previous step, as in a perturbation analysis, choose *Modify*.

Applying Constraints

Constraints can also be applied as either a new constraint set, or to modify current constraints. If you have defined a kinematic constraint set in the *Model Write* dialog box with boundary conditions that you would like to remain on your model, choose *Modify*. However, if you would like to define completely new constraints, choose *New*. For modal superposition analysis types such as *MODAL DYNAMIC, and *STEADY STATE DYNAMICS (not Direct), this option is ignored since constraints cannot be modified or created in a modal superposition method. Furthermore, any constraints defined in a newly chosen constraint set for this *STEP will be ignored.

Note: This option will be used for all *BOUNDARY conditions in the ABAQUS *STEP, including certain FEMAP nodal loads such as enforced displacements, velocities, and accelerations.

Procedure Definition

The options in this group allow you to choose the analysis type to be performed. Remember that you will not be able to change your previous analysis type selection from structural to heat transfer types (or vice versa).

Type

Your choice here indicates the type of analysis that you want to perform. If you chose a structural STANDARD analysis type, you may select between *Static, Modes, Transient Response/Time History, SS Dynamics/Frequency Response, Buckling, Explicit Transient Visco (Creep)*. If you chose a heat transfer type previously, you may choose between *Steady State* and *Transient Heat Transfer* types. You can perform any of the current analysis types, even different types in different steps. If you previously chose *Explicit Transient Dynamics*, this will be the only option available to you.

Depending upon which analysis type you choose, some of the following options must also be specified:

Time Period and Increments

These options are translated directly to the second card of the *STATIC, *DYNAMIC, *MODAL DYNAMIC, and *HEAT TRANSFER (both steady state and transient) procedures. They control the way that ABAQUS will increment through the time step.

For most analyses, you need to:

- define the *Time Period* as the total time for the analysis
- define the *Initial Time Increment* for the analysis
- define the *Min Time Increment* that can be used in automatic cutback
- define the *Max Time Increment*.

When translating for ABAQUS EXPLICIT, you should typically only define the *Time Period* and *Scale Factor*. You can specify a *Max Time Increment*, but this may cause the run to fail in many cases.

Note: For some analyses, you can also use tolerance control. For the detailed procedure, see Section 8.2.1.2, "Analysis Process Overview".

Number of Eigenvectors/Points, Min, Max and Shift Frequency

This information is written to the second card of the *FREQUENCY, *BUCKLE, and *STEADY STATE DYNAMICS options.

- For the *FREQUENCY procedure, ABAQUS will extract frequencies until it reaches either the number you specify or the *Max Frequency* limit.
- For a normal modes analysis, specify the *Max Frequency* and the *No. of Eigenvalues/Points*. You can also specify a *Shift Frequency* if you do not want to recover frequencies near 0 Hz.
- For frequency/harmonic response analyses, you can specify the number of points to be used in the analysis, as well as the minimum and maximum frequencies. These values are translated directly to the second card of the *STEADY STATE DYNAMICS option.

For additional frequencies, you can use a function when defining the load set options for dynamic analysis. For the detailed procedure, see Section 8.2.1.1, "Preparing the Model for Analysis".

NonLinear Geometry

If checked, this option adds the NLGEOM option to the *STEP command. This will account for geometric non-linearity during this and subsequent steps, and is only relevant for stress analysis. This option is required for models which use hyperelastic materials. FEMAP will recognize that hyperelastic materials exist in the model, and then check this option. If this option is checked when you enter FEMAP, deselecting it will almost always cause your analysis to fail.

Linear Perturbation Step

If you check this option, the PERTURBATION option will be added to *STEP, indicating that the current step represents a change from the previous step (as opposed to an independent load step).

Requesting Output

Here you can choose the output that will be calculated for each step. The first choice is whether output should go to the print file, post-processing file, or both. If you plan to use FEMAP for post-processing, you must choose the *Post* option (or *Both*). The print frequency can also be controlled, but must be input as part of the load set for the nonlinear analysis.

The various output types are self-explanatory.. Be aware, however, ABAQUS does not produce all types of output for every element or analysis type. If you check an option but do not get output, that is probably the reason.

Finally, you can choose to recover either integration point or nodal output for elemental data. FEMAP will automatically recover centroidal data if you request any elemental output, simply because centroidal values are required for post-processing.

Note: The *Write Zero Increment* option allows you to write an initial condition step to the output which can be useful when performing multi-set animation. This option, however, is only available in ABAQUS 5.6+.

Include File...

...allows you to input a file into the current *.inp file that you are writing to include information which may not directly supported by FEMAP. This file will be placed at the end of the previous selection. You can select a file before you even write a step to include information in the model portion of the *.inp file. If you would like to include a file inside a step, simply select the step, press OK, and then press the *Include File* button. The file will then be written before the *END STEP card. If *All Load Sets as Individual Steps* is chosen, FEMAP will only write the file at the end of the Last STEP written. It will not include the file in all STEPs. Therefore, the I*nclude File* option should rarely be used with the *Load Sets as Individual Steps* option.

You can even use this option to define a full step for analyses not currently supported by FEMAP. If this file is included before the first step, you define the step in your file, including the *STEP option as the first line and the *END STEP as the last line. If steps have already been defined, you must start the file with an *END STEP to complete the previous step, and then define the *STEP information. The *END STEP should not be included in this case at the end of the file because FEMAP will automatically write this card.

Specifying Additional Analysis Steps

To write the step that you have just defined to the file, press *OK*. You will notice a momentary flash of the dialog box, and the window title will change to reflect the new step number that you are defining. You can repeat this process as many times as necessary, selecting new loads, constraints, or other options. After writing the final step (by pressing *OK*), press *Cancel* to complete the process and close the ABAQUS input file.

8.2.2.3 Overrides/Group Contact

This section describes element formulation overrides as well as group (old) contact. It is important to note that all element formulations can be chosen in FEMAP with the *Formulation* option under the *Element/Property Type* dialog box (under *Model, Property*), or the *Modify, Update Elements, Formulation* command. The options in this particular dialog box cause the element formulations to be overridden by the chosen option.

Note: Group contact (old) was made obsolete by inclusion of contact segments and pairs directly in FEMAP.

This version of the *ABAQUS Model Write* dialog box allows you to specify options that control how your model will be written for both structural and heat transfer analysis types. Structural analysis types utilize all these dialog options, while heat transfer analyses utilize only *Title*, *Initial Conditions*, and *Write All Groups as Sets.*

For information on the options that options are repeated from the main *ABAQUS Model Write* dialog box, see Section 8.2.2.1, "Starting to Export".

Line Elements

These options allow you to choose the element formulation that will be used when FEMAP line (rod, bar, beam...) elements are written to your ABAQUS file. FEMAP will read the types of elements in your model file, and will either use ABAQUS planar (2-D) beams, trusses, etc. if plane strain or axisymmetric FEMAP element types are present, or (3-D) beams and trusses in space if plane strain or axisymmetric element types are not present.

If you choose the *Hybrid Elements* option, FEMAP will choose the appropriate hybrid type (T3D2H, B31H, PIPE31H...). If you check *Cubic Beams*, FEMAP bar and beam elements will be translated to B23 or B33 (B23H or B33H with hybrid selected), instead of the B21, B31 elements. FEMAP rod elements which where formerly translated as C1D2, C1D2H elements are now translated as T2D2, T2D2H, T3D2, and T3D2H ABAQUS Truss elements. The C1D2 series will only be output if *pre-Version 5.4 Format* is checked.

Note: When performing an axisymmetric or plane stress/strain analysis, all beam orientation vectors will be written as (0, 0, -1), the negative Z axis, as required by ABAQUS. Be careful to define your inertias based on this orientation.

Shell and Membrane Elements

These options allow you to choose the element formulation that will be used when FEMAP planar/shell (membrane, plate, laminate,...) elements are written to your ABAQUS file. Each option has specific effects on the writing of certain elements. These options are each discussed briefly below. Linear triangular elements have the most available options due to the capability of ABAQUS to use numerous degenerate quadrilaterals as triangular elements. For this reason, a summary table is provided for the FEMAP triangular planar/shell element in the following table

ABAQUS Element	Reduced Integration	5 DOF	Thick Shell	Flat Triangles	pre-Version5.4 Format
STRI35	OFF	N/A	OFF	OFF	N/A
STRI3	OFF	N/A	OFF	ON	N/A
S3R	OFF	N/A	ON	N/A	N/A
S4R	ON	OFF	N/A	N/A	OFF
S4R5	ON	ON	N/A	N/A	N/A
S4RF	ON	OFF	N/A	N/A	ON

- The *Thick Shell Behavior* option has an effect on linear triangular, linear quadrilateral, and parabolic quadrilateral elements. If *Reduced Integration* is off and *Thick Shell Behavior* is on, linear triangles will be written as S3R elements instead of STRI3 or STRI35. Quadrilateral linear elements will be written as S43 elements with "Thick Shell" on, instead of S4R5 or S4RF elements, and parabolic quadrilateral elements will be written as S8R instead of S8R5 elements.
- *Reduced Integration* controls the writing of triangular elements as degenerate quadrilateral elements. It has no effect on true quadrilateral elements. If *Reduced Integration* is selected, all triangular elements will be written as degenerate quadrilaterals (S4R, S4R5, or S4RF, and S8R5). If this option is not selected, triangular elements will be written (S3R, STRI3, or STRI35, and STRI65).
- Choosing the *5 DOF Elements* option enables you to choose S4R5 elements instead of S4R or S4RF elements, as well as S8R5 elements instead of S8R elements for parabolic quadrilaterals. The final option in this section, *Flat Triangles*, enables you to write linear triangular elements as STRI3 elements instead of STRI35 elements. This option has no impact unless neither the *Reduced Integration* nor *Thick Shell* options are chosen.

Note: The S4RF elements were combined with the S4R elements in ABAQUS v5.4. These elements are only written when *pre-Version 5.4 Format* is checked, and neither the *5-DOF Elements* nor *Thick Shell* options are selected. If neither of these options are chosen and *pre-Version 5.4 Format* is not checked, linear quadrilateral elements will be written as S4R elements.

In addition to choosing the element formulations, you can also set the number of integration points to be used with planar elements. If you leave this value blank, nothing is written and the defaults for each element type are used. If you specify a value, it is written to the *SHELL SECTION commands.

Solid Continuum Elements

Just like for line and shell elements, checking *Hybrid Elements* tells FEMAP to select elements that use a hybrid formulation (CPxxH, CExxH, CAXxxH and C3DxxH). In addition, you can select from the *Standard*, *Incompatible Modes* and *Reduced Integration* formulations. *Incompatible Modes* chooses the "I" series (CPxxI, C3DxxI...) elements, when available. *Reduced Integration* chooses the "R" series (CPxxR, C3DxxR...) elements, when available.

You may also choose special modified contact elements for parabolic triangular plane stress/plane strain elements (CPx6M...) and parabolic solid tetrahedral (C3D10M and C3D10MH) elements. These elements are only available in ABAQUS v5.6 and higher

Since FEMAP has no 2D plane stress element, you must use plane strain elements, even when trying to model plane stress problems. Then, to represent plane stress, check the *Plane Strain as Plane Stress* option, and FEMAP will use CPSxx elements instead of CPExx elements.

Pre-Version 5.4 Format

This option allows you to select a format for the input file that is compatible with versions prior to ABAQUS 5.4. This option generates changes involving S4RF elements, truss elements, membrane elements, rigid surfaces, and rigid bodies,. The element S4RF was combined with element S4R in version 5.4 and is only written if *pre-Version 5.4 Format* is checked. Truss elements (TnD2 series) now replace the C1D2 series linear link elements. Membrane elements now have their own *MEMBRANE property card for ABAQUS version 5.4. If the pre-Version 5.4 format is selected, *SOLID SECTION is used as the property option for membrane elements.

If the *Pre-Version 5.4 Format* button is not checked, rigid surfaces and rigid bodies can be modeled with the ABAQUS *CONTACT PAIR approach. If this button is selected, rigid bodies and contact pairs should not be used, and special interface elements will be generated to model contact with rigid surfaces.

For more information, see "Defining a Rigid Surface".

Rigid Surfaces...

...lets you select a FEMAP group which will be translated as a rigid surface, and the associated slave surface, contact node set, or interface elements.

For more information, see "Defining a Rigid Surface".

Slide Lines...

...lets you select a FEMAP group which will be translated as a slide line, and the associated interface elements.

For more information, see "Defining a Slide Line".

Rigid Bodies...

...lets you select a FEMAP group which will be translated as a rigid body, and the associated slave surface or contact node set.

For more information, see "Defining a Rigid Body".

Contact Pairs...

Pressing this button will allow you to utilize FEMAP groups to define a contact pair.

For more information, see "Defining a Contact Pair".

Important Note Regarding Group Contact Modeling

You should review the restrictions concerning contact modeling in ABAQUS. FEMAP performs many checks in an attempt to prevent you from violating these rules. Even though these checks are quite extensive, they are not all inclusive. Therefore, you should carefully define the elements in the surfaces of the contact pair.

In addition, FEMAP has three restrictions in addition to those maintained by ABAQUS:

- Structural and continuum elements may not be included in the same slave surface. If this type of contact is required, two separate contact situations must be created.
- Structural elements will automatically be written with their positive normal (SPOS or S1) as the contact side. You must create the model in this manner, or reverse the normal direction using either *Modify, Update Element, Reverse* or *Tools, Check, Normals* commands. If contact must be maintained on both sides of these elements, you must either create two sets of elements with opposite normals, or edit the input file, changing the appropriate SPOS (or S1) to SNEG (or S2).
- Continuum elements must be a subset of the exposed surfaces of the model.

FEMAP will also utilize the ABAQUS defaults for sliding type and trimming. Finite sliding will be used as the default, while small sliding will be chosen when finite sliding is not available (such as contact between 3-D deformable bodies). Automatic trimming will also be utilized for slave surfaces containing continuum elements. Master surfaces with continuum elements will not be trimmed. You must modify these options manually if different options are required.

Defining a Rigid Surface

Although FEMAP does not have the equivalent of an ABAQUS rigid surface, the interface can use a FEMAP group to create one. First, however, you must properly construct your model and the group that you will use. The entities of the group will differ based upon the *Pre-Version 5.4 Format* button.

ABAQUS v5.4 or Above (Contact Pair Approach)

To successfully create a rigid surface, a FEMAP group must contain the following entities:

- either one or more lines or arcs, or a surface
- one reference node
- one or more FEMAP elements or nodes defining the slave surface
- optionally, a single GAP property

Lines, Arcs, and Surfaces

If you select lines and/or arcs, they must form a continuous path. You must select the lines such that as you would walk along the lines, the contact is on your left. This can be accomplished by using *View Options, Tools and View Style, Curve and Surface Accuracy* to show the direction of the lines. The first line selected must be the start of the rigid surface, and it must be in the proper direction. FEMAP will automatically form the remaining lines in the appropriate direction.

The lines and arcs will be converted to a segment (if there are plane strain or axisymmetric elements in the model) or CYLINDER rigid surface. Splines may not be chosen, and if you choose arcs, they must always have an included angle less than 180 degrees. If you need larger arcs, break the arc into pieces before you export.

If you select a FEMAP surface in the group, it will be converted to a rigid BEZIER surface. The number of surface divisions that are set for the display of the surface will be used to create the VERTEX and PATCH commands for the BEZIER surface. If you want a better surface definition, just increase the number of surface divisions. Although the BEZIER surface with VERTEX and PATCH options are no longer documented in ABAQUS v5.4 and higher, they are still available. However, for most contact problems, it is recommended that the RIGID BODY approach be utilized.

Reference Node

You must also select a reference node for the rigid surface. This node can (and usually should) be constrained in your model. The motion, or lack thereof, of this reference node, will determine the motion of the rigid body.

Elements and Nodes

The final required contents of the group are either one or more elements or nodes defining the slave surface. If elements are present, FEMAP will attempt to write these elements as a slave surface. If no elements are in the group, FEMAP will assume that the slave surface is defined with a contact node set. The first node selected in the group is always used as the reference node for the rigid surface. This node will not be written as part of the contact node set. All nodes selected in this group after the first (reference) node will be written as part of the contact node set. FEMAP will only write a contact node set if there are no elements contained in the group.

GAP Property

The GAP property is used to control the *SURFACE INTERACTION command. Out-of-plane thickness or cross-sectional area will be written on the *SURFACE INTERACTION card if specified in the selected GAP property. Friction properties can also be specified (including ANISOTROPIC) on the GAP property, and will be written on the *FRICTION command. If you are performing 3-D contact with plates, you will also have to supply an interface normal on the GAP property so FEMAP can properly orient the 3-D surface.

pre-ABAQUS v5.4 (Interface Element Approach)

To successfully create a rigid surface, a FEMAP group must contain the following entities:

- either a surface, or one or more lines and/or arcs
- one or more FEMAP line or plane plot-only elements
- one reference node
- optionally, a single GAP property

Surfaces, Lines, and Arc

If you select a FEMAP surface in the group, it will be converted to a rigid BEZIER surface. The number of surface divisions that are set for the display of the surface will be used to create the VERTEX and PATCH commands for the BEZIER surface. If you want a better surface definition, increase the number of surface divisions. If instead you choose lines and/or arcs, they must form a continuous path. They will be converted to a SEGMENT (if there are plane strain or axisymmetric elements in the model) or CYLINDER rigid surface. Splines may not be chosen, and if you choose arcs, they must always have an included angle less than 180 degrees. If you need larger arcs, break the arc into pieces before you translate.

Plot-Only Elements

In addition to the geometry, you must create and select plot-only elements along the edge/face where you want to check for contact. These are then translated to IRSxx interface elements.

Reference Node

Next, you must select a reference node for the rigid surface. This node can (and usually should) be constrained in your model. It is referenced by all of the interface elements.

GAP Property

The GAP property is used to control the *INTERFACE command. If you specify a coefficient of friction (Y direction) on this GAP property, it will be written as a *FRICTION command.

Defining a Slide Line

The process to define a slide line is very similar to that for a rigid surface. In this case, however, the group must contain:

- one or more plot-only line elements
- an ordered series of nodes to represent the slide line
- a GAP property

The plot-only line elements are converted to interface (ISLxx) elements, and the nodes are written as the slide line. You must be careful to select these items into the group in the proper order. FEMAP translates them in the order that you select them.

Just as for rigid surfaces, the GAP property is used for friction on the *INTERFACE command.

Note: When selecting items into the group, you should always use the "ID" rules. Selections using other methods do not allow FEMAP to determine the proper ordering for the output and will be ignored.

Defining a Rigid Body

Similar to rigid surface contact, FEMAP will allow you to create ABAQUS rigid bodies by selecting a FEMAP group. The procedure is very similar to rigid surface definition, except one or more FEMAP line or plane plot-only elements must be contained in the group to define the rigid body, instead of lines and arcs.

To successfully create a rigid body, a FEMAP group must contain:

- one or more FEMAP line or plane plot-only elements
- one reference node
- one or more FEMAP elements or nodes defining the slave surface
- optionally, a single GAP property

Plot-Only Elements

The plot-only elements are utilized to define the rigid body. Either line or plane plot-only elements must be selected, but not both. Planar elements will be translated to R3D3 and R3D4 elements, while line elements will be translated to R2D2 (if plane strain elements are present), RAX2 (if axisymmetric elements are present), or RB3D2 elements. These elements must be created such that the positive faces of the element define the contact surface. FEMAP will automatically choose the positive normal to define the master contact surface. All plot-only elements contained in the group will be written as rigid elements, referencing the rigid body.

Reference Node

The reference node, slave surface, and GAP property are all chosen similar to the rigid surface definition.

For more information, see "Defining a Rigid Surface".

Rigid Elements

Rigid elements can only be used in one rigid body definition. However, this rigid body can be put in contact with multiple surfaces. To create rigid body contact between several surfaces and the same rigid body, define a group for each contact pair, and select the same reference node and plot-only elements in each group. You must take care to select the same plot only elements because FEMAP will assume a new rigid body is being defined if the element selection is not identical, and errors will result. Once the plot-only elements and reference node are appropriately selected, you may then select different elements or nodes in each group to define different slave surfaces. FEMAP will generate the slave surfaces and put them in contact with the rigid body.

Defining a Contact Pair

FEMAP also has the capability to define contact pairs between deformable elements. To successfully create a contact pair, you must create:

- one group containing the elements of the master surface
- a second group containing the elements or nodes of the slave surface
- optionally, a GAP property containing width/area and/or friction information

When you select the *Contact Pair* button, you will be prompted to input the group of the master surface, followed by a prompt for the slave surface, and finally the property to define the *SURFACE INTERACTION and *FRICTION options.

8.2.3 Performing an ABAQUS Analysis

Once you have written the ABAQUS input file, you can analyze it with ABAQUS as you would with any other data file.

Alternatively, if you are running both FEMAP and ABAQUS on a Windows operating system, you can have FEMAP automatically launch ABAQUS, run the solution, and recover the results. In this case, before you start

FEMAP, you must first define an environment variable in Windows. Name the variable ABAQUS_EXE, give it a value equal to the complete file/path name of the ABAQUS executable. For example:

```
ABAQUS_EXE = c:\abaqus\v61\abaqus.exe
```

Once this variable is set, FEMAP will automatically launch ABAQUS when you press *Analyze* from the *Analysis Set Manager*, or once the input file is written from *File, Export*. FEMAP will monitor the analysis and return results when it is finished.

8.2.4 Reading ABAQUS Models

Just as you can translate a FEMAP model to ABAQUS, you can also read a ABAQUS model into FEMAP. From the ABAQUS input file (*.inp), FEMAP can read node and element connectivity along with most material, property, load, constraint and contact definitions. FEMAP can also read the *INCLUDE command, which references external files that include other model data.

To read in a model, pick *File, Import, Analysis Model*, then choose the input file you want to read. If * INCLUDE commands are present in the file, FEMAP will also read those files specified in the path.

8.2.5 Post-processing ABAQUS Results

When you have completed your ABAQUS analysis, you can load results into FEMAP for post-processing. You must also start with a FEMAP model that corresponds to the ABAQUS analysis.

FEMAP reads the ASCII formatted ABAQUS Post File (*.FIL). You must include a *FILE FORMAT, ASCII command, along with *NODE FILE, *EL FILE, and *CONTACT FILE commands in your model to produce this file. As long as you built your model in FEMAP, and chose the Post option when you translated, these commands will be added automatically.

Note: If you have an ABAQUS EXPLICIT binary results file (*.res), you will need to run the ABAQUS convert utilities to convert it to a *.fil file. This, however, is still a binary file, which must then be converted to an ASCII file (which will have the extension *.fin) using utilities provided with ABAQUS.

To load the results, choose the *File, Import, Analysis Results* command, and select *ABAQUS*. FEMAP will display the standard file access dialog box so you can choose the file that you want to read. Before reading data from a file, you will see a brief description of the file in the *Messages* window, and you will be asked to confirm that this is the file you want to read. FEMAP will then read the output data.

Output for Post-processing

The formats and contents of the ABAQUS files are described in the ABAQUS User’s Manual. From the Post file, FEMAP will read nodal displacements, velocities, accelerations, reaction forces, and loads. In addition, FEMAP reads elemental temperatures, stress components, stress invariants, section forces and moments, strain energy, and total strain components. These output types are supported data. FEMAP also has the capability to read many ABAQUS output types as unsupported data, including nodal temperatures, nodal heat fluxes, and elemental heat fluxes.

From the values that are read, FEMAP will also compute magnitudes of the nodal values (total translation, total rotation, etc.). Likewise, if they were not read from the file, FEMAP computes principal, max shear, mean, and Von Mises stresses whenever possible.

8.3 ANSYS Interfaces

FEMAP provides direct interfaces to the ANSYS PREP7 and post-processing file formats. You can write a FEMAP model to the ANSYS PREP7 format for analysis, read an existing ANSYS model, or read analysis results for post-processing.

Topics in this section include:

- Section 8.3.1, "Writing an ANSYS Model with Model, Analysis"
- Section 8.3.2, "Writing an ANSYS Model with File, Export" (Obsolete)
- Section 8.3.3, "Performing an ANSYS Analysis"
- Section 8.3.4, "Reading ANSYS Models"
- Section 8.3.5, "Reading ANSYS Analysis Results"

For more information on the entities that are translated, see:

- Section 7.1, "Translation Table for ANSYS, I-DEAS, NASTRAN, and MSC Patran"

8.3.1 Writing an ANSYS Model with Model, Analysis

The *Model, Analysis* command opens the *Analysis Set Manager*, which enables you to write a FEMAP model to ANSYS. The interface supports ANSYS versions 4, 5.x -9.0. Alternatively, you can also use the *File, Export, Analysis Model* command (see Section 8.3.2, "Writing an ANSYS Model with File, Export").

This section includes the following topics:

- Section 8.2.1.2, "Analysis Process Overview" using the *Analysis Set Manager*
- Section 8.3.1.2, "Preparing the Model for Analysis" - guidelines on preparing for each analysis type
- Section 8.3.1.3, "Setting the ANSYS Analysis Parameters" - descriptions of the ANSYS interface dialog boxes

8.3.1.1 General Analysis Process Overview

The *Analysis Set Manager* lets you create an analysis set, which is an input file for ANSYS. To define analysis set:

1. Pick *Model, Analyze* to open the *Analysis Set Manager* dialog box.
2. Pick the first item on the list, then pick *New*. (You can also double-click on the item).
3. On the *Analysis Set* dialog box, choose *5..ANSYS* as the *Analysis Program*. Select the *Analysis Type: 1..Static*, *2..Normal Modes/Eigenvalue*, *3..Transient Dynamic/Time History*, *4..Frequency/Harmonic Response*, *6..Random Response*, *7..Buckling*, *10..Nonlinear Static*, *12..Nonlinear Transient Response*, *20..Steady-State Heat Transfer*, or *21..Transient Heat Transfer*. These options determine which element types (structural or heat transfer) will be written, and set the defaults for the other ANSYS interface dialog boxes. (With dynamic and random response analysis types, you will see some additional dialog boxes. See Section 8.3.1.2, "Preparing the Model for Analysis".)

   You can also choose to use a *Linked Solver* or *VisQ*. See Section 2.6.2.6, "Solvers" or Section 4.10.2.1, "Run Analysis Using Linked Solver / VisQ / Local Settings" in the FEMAP Commands Manual for more information.
4. Pick *Next* to open the *ANSYS Executive and Solution Control* dialog box. (Alternatively, you can pick *OK* to close the *Analysis Set* dialog box. From the analysis set list, you can then double-click on an option to bring up the dialog box where the option is defined.) Contains information about current ANSYS Version setup to run with FEMAP and allows you to specify a number of command line arguments.
5. Pick *Next* to open the *ANSYS Model Write* dialog box. Specify solve options.
6. Pick *Next* to open the *Master Requests and Conditions* dialog box. The master requests and conditions are the default boundary conditions. Enter a title.
7. Pick *Next* to open the *Boundary Conditions* dialog box. Select the constraints and loads.
8. For some analysis types (*Static, Normal Modes/Eigenvalue, Nonlinear Static*), you can create cases. Each case includes a title, options (depending on analysis type), and boundary conditions. In the *Analysis Set Manager*, double-click on *No Cases Defined*. Pick *Next* to work through the dialog boxes to create the case.

For more information, see also:

- Section 8.3.1.2, "Preparing the Model for Analysis" for how to work with each analysis type
- Section 8.3.1.3, "Setting the ANSYS Analysis Parameters" for descriptions of the ANSYS interface dialog boxes

8.3.1.2 Preparing the Model for Analysis

This section describes the steps in preparing for each analysis type. It includes these topics:

Static Analyses:

- "Preparing for a Static Analysis"
- "Preparing for a Nonlinear Static Analysis"

Dynamic Analyses:

- "Preparing for a Normal Modes/Eigenvalues Analysis"
- "Preparing for a Transient Dynamic/Time History Analysis"
- "Preparing for a Nonlinear Transient Analysis"
- "Preparing for a Frequency/Harmonic Response Analysis"
- "Preparing a Random Response Analysis"
- "Preparing for Buckling Analysis"
- "Notes on ANSYS Dynamic Analyses"

Heat Transfer Analyses:

- "Preparing for Steady-State Heat Transfer Analysis"
- "Preparing for Transient Heat Transfer Analysis"

Other Topics:

- "Preparing for Nonlinear Contact Analysis"
- "Special Cases"

For more information, see also:

- Section 8.2.1.2, "Analysis Process Overview" for information on how to use the *Analysis Set Manager*
- Section 8.3.1.3, "Setting the ANSYS Analysis Parameters" for descriptions of the ANSYS interface dialog boxes

Preparing for a Static Analysis

To set up a static analysis in ANSYS:

1. Build and mesh the FEMAP model. Apply constraints and loads.
2. Set up the analysis with the FEMAP *Analysis Set Manager*.

   As part of this process, you can define cases to perform multiple analyses with different combinations of load and/or constraint sets.

Preparing for a Nonlinear Static Analysis

To set up a nonlinear static analysis in ANSYS:

1. Build and mesh the FEMAP model. Apply constraints.
2. Define nonlinear load sets in FEMAP. For the first nonlinear load set, you must define nonlinear analysis options. Pick *Model, Load, Nonlinear* to set the *Load Set Options for Nonlinear Analysis*. For details, see Section 4.3.5.1, "Model, Load, Nonlinear Analysis..." in *FEMAP Commands*.

   Once you define the nonlinear load set options for the first load set, the interface will use these options for the remaining load sets.
3. Set up the analysis with the FEMAP *Analysis Set Manager*.

As part of this process, you can define cases to perform multiple analyses with different combinations of load and/or constraint sets.

Preparing for a Normal Modes/Eigenvalues Analysis

To set up a normal modes/eigenvalues analysis in ANSYS:

1. Build and mesh the FEMAP model. Apply constraints and loads.
2. Set up the analysis with the FEMAP *Analysis Set Manager*.

   As part of this process, you can define cases to perform multiple analyses with different combinations of load and/or constraint sets.

   You can also control the master degrees of freedom and number of modes or frequency range with the *ANSYS Dynamic Analysis Options* dialog box.

Preparing for a Transient Dynamic/Time History Analysis

To set up a transient dynamic/time history analysis in ANSYS:

1. Build and mesh the FEMAP model. Apply constraints.
2. Define a dynamic load set in FEMAP. Pick *Model, Load, Dynamic* to set the *Load Set Options for Dynamic Analysis*. The options that you pick depend on whether you want a direct or modal analysis.

   For a direct analysis, set the *Solution Method* to *Direct Transient*.

   For a modal analysis, set the *Solution Method* to *Modal Transient*. Under *Response Based on Modes*, enter the number of modes and lowest and highest frequencies.

   For details, see Section 4.3.5.2, "Model, Load, Dynamic Analysis..." in *FEMAP Commands*.
3. Set up the analysis with the FEMAP *Analysis Set Manager*.

   If the modal analysis method is active, you will be able to define additional parameters *ANSYS Dynamic Analysis Options* dialog box. These parameters include master DOF, number of modes to extract, and the lowest and highest frequencies (ANSYS command MODOPT).

Note: To prevent translation errors, be careful to set appropriate options for both the dynamic load set and the analysis set. The interface won't write the load set to the ANSYS solver if the *Solution Method* is turned off. It also won't write the load set if the *Solution Method* (for example, *Modal Frequency*) doesn't match the *Analysis Type* for the analysis set (for example, *Transient Dynamic/Time History*).

Preparing for a Frequency/Harmonic Response Analysis

To set up a frequency/harmonic response analysis in ANSYS:

1. Build and mesh the FEMAP model. Apply constraints.
2. Define a dynamic load set in FEMAP. Pick *Model, Load, Dynamic* to set the *Load Set Options for Dynamic Analysis*. The options that you pick depend on whether you want a direct or modal analysis.

   For a direct analysis, set the *Solution Method* to *Direct Frequency*.

   For a modal analysis, set the *Solution Method* to *Modal Frequency*. Under *Response Based on Modes*, enter the number of modes and lowest and highest frequencies.

   For details, see Section 4.3.5.2, "Model, Load, Dynamic Analysis..." in *FEMAP Commands*.
3. Set up the analysis with the FEMAP *Analysis Set Manager*.

   You can define solution frequencies to be expanded (ANSYS command NUMEXP) on the *ANSYS Dynamic Analysis Options* dialog box.

Note: To prevent translation errors, be careful to set appropriate options for both the dynamic load set and the analysis set. The interface won't write the load set to the ANSYS solver if the *Solution Method* is turned off. It also won't write the load set if the *Solution Method* (for example, *Modal Frequency*) doesn't match the *Analysis Type* for the analysis set (for example, *Transient Dynamic/Time History*).

Preparing for a Nonlinear Transient Analysis

Preparing for nonlinear transient analysis is nearly identical to transient dynamic/time history analysis (see "Preparing for a Transient Dynamic/Time History Analysis"), except that:

- the modal transient solution method is not available. If you select the modal transient solution method (on *Load Set Options for Dynamic Analysis*), the load set will not be written to the ANSYS input file.
- nonlinear analysis options must also be active, or the load set will not be written to the ANSYS input file

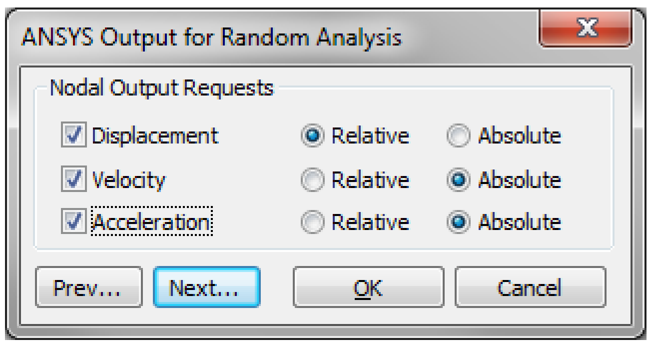
  For the first nonlinear load set, you must define nonlinear analysis options. Pick *Model, Load, Nonlinear* to set the *Load Set Options for Nonlinear Analysis*. For details, see Section 4.3.5.1, "Model, Load, Nonlinear Analysis..." in *FEMAP Commands*.

  Once you define the nonlinear load set options for the first load set, the interface will use these options for the remaining load sets since ANSYS doesn't allow for modifications during the solution process.
- you can choose to activate *Large Deformation Effects* on the *Analysis Set* dialog box

Preparing a Random Response Analysis

Preparing for random analysis is identical to Frequency/Harmonic Response analysis (see "Preparing for a Frequency/Harmonic Response Analysis"), except that:

- the *ANSYS Dynamic Analysis Options* dialog box always appears because a modal analysis is required before every random analysis.



- you can select multiple load sets. All displacement values in the chosen load sets will be exported as base excitations (D), while all forces will be applied as nodal excitations (F).
- you can define output for a random analysis with the *ANSYS Output for Random Analysis* dialog box. This dialog box will appear when you pick *Next* on the *ANSYS Model Write* dialog box.

Note: Imported nodal results from a random analysis are read “as is” from ANSYS Results files. The results are not transformed into the global coordinate system like nodal results from other types of analysis. Therefore, any command in FEMAP which “transforms” nodal output will produce invalid output values for the nodal output vectors in these sets..

Preparing for Buckling Analysis

To prepare for a buckling analysis:

1. Build and mesh the FEMAP model. Apply one constraint set and one load set.
2. Set up the analysis with the FEMAP *Analysis Set Manager*.

   Use the *ANSYS Dynamic Analysis Options* dialog box to enter the number of modes to be extracted and whether elemental results should be calculated.

Notes on ANSYS Dynamic Analyses

This topic reviews the limitations on transient and frequency response loading conditions in ANSYS. Loads on the same node or elemental face must have the same time history (transient) or same phase (frequency). If they do not, FEMAP will write the loading conditions assuming the last time history/phase, and you will most likely get results for a different loading condition than desired.

Also, with regard to Rayleigh damping, you can define alpha and beta damping for frequency analysis by defining the overall structural damping coefficient (G) and in some cases, the frequencies for element and system damping.

For frequency and random response analyses, FEMAP will automatically compute alpha and beta. If only one frequency is defined for the analysis, FEMAP assumes alpha is 0, and computes beta from G/wi where wi is the frequency. If a range of frequencies are specified, FEMAP will compute alpha and beta based on the equation

```
G = alpha/wi + beta*wi
```

by assuming G is constant over the frequency range. Two simultaneous equations are produced at the two frequencies which define the range.

For Rayleigh damping in transient analysis, FEMAP uses the *Frequency for System Damping (W3 - HZ)* and *Frequency for Element Damping (W - Hz)* input on the *Load Set Options for Dynamic Analysis* dialog box (*Model, Load, Dynamic Analysis)* to compute alpha and beta damping values. Alpha is simply the product of the overall structural damping coefficient (G) and W3 (*Frequency for System Damping*). Beta is G divided by W4 (*Frequency for Element Damping*). If G is zero, Rayleigh damping is ignored. Also, if W3 or W4 is zero, alpha or beta, respectively, will not be written.

Preparing for Steady-State Heat Transfer Analysis

Preparing for a steady-state heat transfer analysis is just like preparing for static analysis (see "Preparing for a Static Analysis"), except that instead of defining nodal constraints, you define nodal temperatures (in a load set) for your boundary conditions.

Preparing for Transient Heat Transfer Analysis

To set up a transient heat analysis in ANSYS:

1. Build and mesh the FEMAP model. Apply constraints.
2. Define time-dependent nodal or elemental thermal loads. To do this, create a function (use *Model, Function*). Set the *Type* to *1. vs Time*, and use the *Data Entry* methods to create the time-dependent thermal load (for example, heat generation vs. time, heat flux vs. time).

   When you create the thermal load (for example, using *Model, Load, Nodal, Heat Generation*), define the load using the function (for example, heat generation vs. time).
3. Set up the analysis with the FEMAP *Analysis Set Manager*. Use the *Nonlinear Control Options* dialog box to define step size and the initial time increment.

   As part of the process, you can define cases to perform multiple analyses with different combinations of load and/or constraint sets.

Preparing for Nonlinear Contact Analysis

Surface-to-surface contact is defined in ANSYS using a combination of contact surface and target segment elements. ANSYS looks for contact only between contact surfaces and target segments that share the same real constant set. To define contact for ANSYS:

- Define the real constant set using the *Model, Contact, Contact Property* command. Pick the *ANSYS...* button to modify the KEYOPTs.
- Define the target segments using the *Model, Contact, Segment Surface* command. The target segments (TARGE169 (2-D) and TARGE170 (3-D)) can be either rigid or deformable. When defining a rigid target, you can use only line elements or Face 1 of a plate element.
- Define the contact surfaces using the *Model, Contact, Segment Surface* command. The contact surfaces can be 2-D (CONTA171 and CONTA172) or 3-D (CONTA173 and CONTA174), and must be deformable.
- Model a contact pair using the *Model, Contact, Contact Pair* command. The dialog box asks for a *Master* (the target segment) and a *Slave* (the contact surface), as well as a *Property* (the shared real constants set).
- Set up the time steps for a nonlinear analysis using the *Model, Load, Nonlinear Analysis* command.

For more information about these commands, see Section 4.4, "Creating Connections and Regions"in *FEMAP Commands*.

Hint: To create a rigid entity, use line elements for 2-D analysis and plate elements with the element normal direction facing the contact for 3-D analysis.

Special Cases

Depending on the contents of your model, you may see additional questions or warnings as the file is translated. For example, ANSYS requires certain IDs, especially for coordinate systems. If your model has entities which are outside of the allowable ID range, FEMAP will ask if you want to renumber prior to translating. Likewise, axisymmetric and other 2-D elements must lie in the global XY plane for ANSYS. If you have built your model in a different global plane, FEMAP will ask if you want to automatically realign it to the XY plane. FEMAP can only “flip” between global planes. You should never build an axisymmetric model in a skewed plane.

For axisymmetric models, you will also have an opportunity to specify a scale factor for loads. This factor is normally 1.0, but can be adjusted depending on whether you specified loads on a per radian or per revolution (360 degrees) basis. This is especially important since the required conventions change between ANSYS Revision 4.4 and 5.0.

8.3.1.3 Setting the ANSYS Analysis Parameters

To set up an ANSYS analysis, you will work through the following dialog boxes:

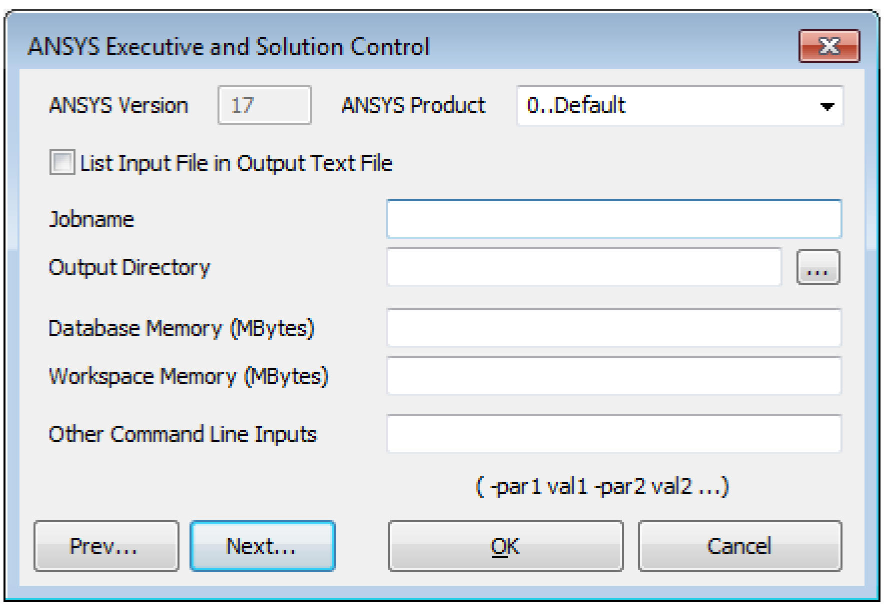
- "ANSYS Executive and Solution Control"
- "ANSYS Model Write"
- "Master Requests and Conditions"
- "ANSYS Dynamic Analysis Options"
- "Boundary Conditions"
- "Cases"

For information on how to prepare the model for each analysis type, see:

- Section 8.3.1.2, "Preparing the Model for Analysis"

ANSYS Executive and Solution Control

The ANSYS Executive and Solution Control dialog box contains information about current ANSYS Version setup to run with FEMAP and allows you to specify a number of command line arguments.



ANSYS Version

The version number displayed relates to the solver executable specified for *Ansys* on the *Solvers* tab of the *File, Preferences* command or by the ANSYS_EXE environment variable. See Section 2.6.2.6, "Solvers" or Section 4.10.2.1, "Run Analysis Using Linked Solver / VisQ / Local Settings" in the FEMAP Commands Manual for more information. If no version is defined, defaults to version 10.

ANSYS Product

Allows you to choose a particular *ANSYS Product* to launch, which may be useful to select to only use the license(s) needed for your analysis. Choose from *0..Default*, *1..ANSYS Multiphysics*, *2..ANSYS Mechanical*, *3..ANSYS Structural*, *4..ANSYS Mechanical Emag*, or *5..ANSYS Mechanical CFL-Flo*. Includes “-p” on command line.

List Input File in Output Text File

When off (default), includes “-b nolist” on command line and does not include the text of the input file in the printed output file. When on, includes “-b list” on command line to list text of input file to printed output file.

Jobname

When specified, is the initial “jobname”, a name assigned to all files generated by the program for a specific model. Includes “-j (specified jobname)” on command line. Has a maximum length of 32 characters and spaces should be avoided. If not specified, assumes file name is the “jobname”.

Output Directory.

When specified, allows you set a location for output to be placed. The “...” icon button can be used to browse to a directory location. Includes “-dir (specified output directory)” on command line. If not specified, uses default output directory for FEMAP.

Database Memory (MBytes)

When specified as a positive value, defines the initial memory allocation for the database. If specified as a negative value, defines a fixed memory allocation for the database throughout the run. Includes “-db (value)” or “-db (-value)” on command line. If no value is specified, “-db” is omitted and default of 1024 MB is used.

Workspace Memory (MBytes)

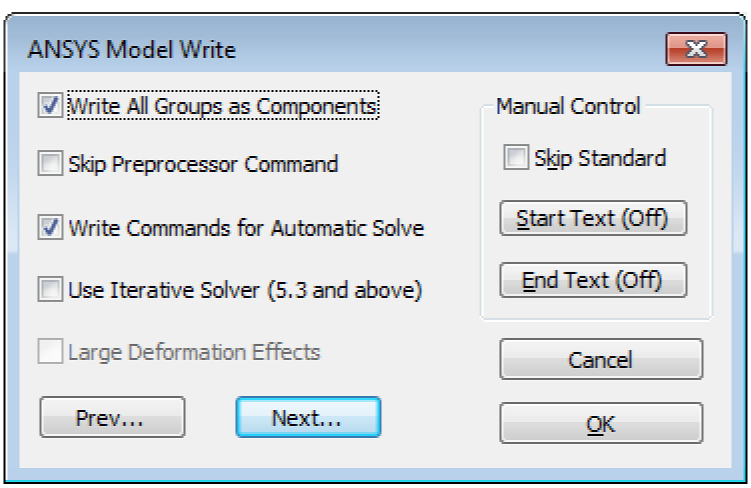
When specified as a positive value, defines the initial overall memory allocation. If specified as a negative value, defines a fixed overall memory allocation throughout the run. Includes “-m (value) or “-m (-value)” on command line. If no value is specified, “-m” is omitted and default of 2048 MB is used.

Other Command Line Options

Allows you to specify other command line arguments. Consult ANSYS documentation for more information.

ANSYS Model Write

The *ANSYS Model Write* dialog box sets up the basic parameters for the analysis. It includes the following sections and fields:



Write All Groups as Components

When enabled, all groups in FEMAP will be written as “components” to the ANSYS input file (default = On). If a group contains both nodes and elements then two separate components will be created, one for the nodes in the group and another for the elements in the group. Each of the components in the ANSYS input file will be automatically generated and follow the same naming convention “CM, G(group #), (NODE/ELEM)”.

Skip Preprocessor Command

When on, the command will cause FEMAP to not write the /PREP7 line to the ANSYS input file (default = Off).

Write Commands for Automatic Solve

If you check this option, FEMAP will write additional commands to the end of your ANSYS file. These commands automatically perform the analysis when you load the file into ANSYS or run the model in batch mode.

Do not select this option if you are running a static, modal, or nonlinear static analysis and want to load the model into ANSYS, and then review it in PREP7 before beginning your analysis.

Use Iterative Solver (5.3 and above)

This option invokes the ANSYS iterative solver, and is only applicable for ANSYS v5.3+.

Large Deformation Effects

For nonlinear analysis (both static and transient), you also have the option to calculate *Large Deformation Effects*.

Manual Control

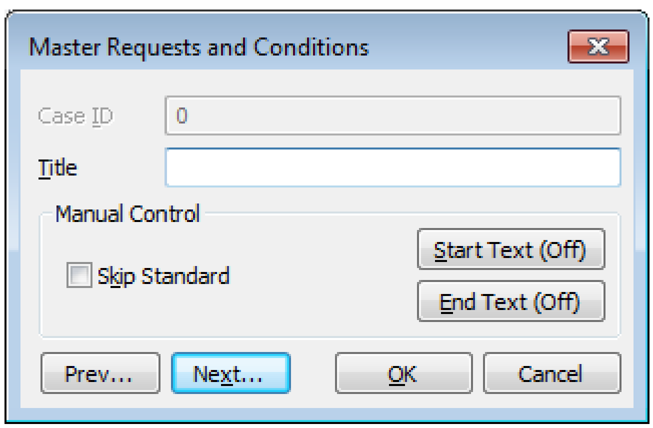
Manual Control options include:

- *Skip Standard*: If this switch is on, the interface will not write the node, element, material, and property data to the ANSYS input file. *Start* and *End Text* will be written to the input file.
- *Start Text*: Pick this option to add text to the beginning of the node, element, material, and property data in the input file.
- *End Text*: Pick this option to add text to the end of the node, element, material, and property data in the input file.

Master Requests and Conditions

In an ANSYS analysis, the master requests and conditions are the default boundary conditions. The analysis will generate one output set for the master requests and conditions, unless you define a case (see "Cases").

On the *Master Requests and Conditions* dialog box, you can enter a *Title* and *Manual Control* options. Once you have entered this data, pick *Next* to set up the boundary conditions.

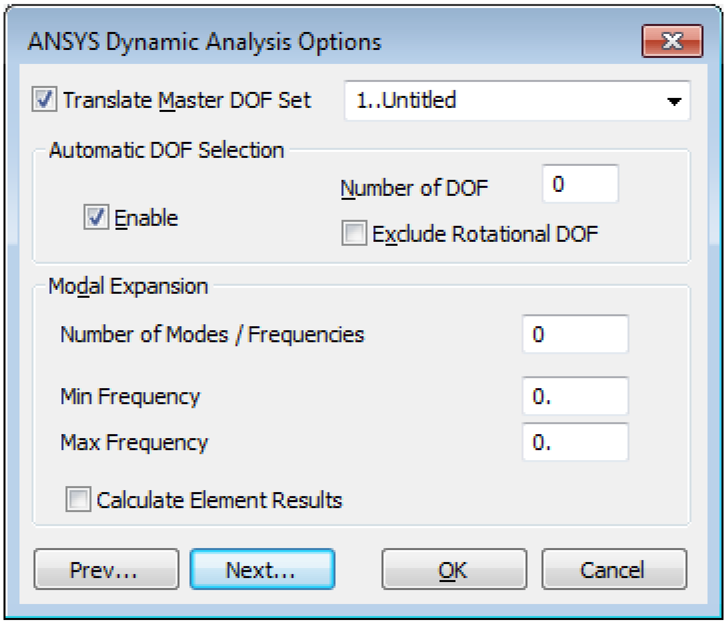


The *Manual Control* options include:

- *Skip Standard*: If this switch is on, the interface will not write the load, constraint, and load option data to the ANSYS input file. *Start* and *End Text* will be written to the input file.
- *Start Text*: Pick this option to add text to the beginning of the load, constraint, and load option data in the input file.
- *End Text*: Pick this option to add text to the end of the load, constraint, and load option data in the input file.

ANSYS Dynamic Analysis Options

If you chose a modal analysis type, you will see the *ANSYS Dynamic Analysis Options* dialog box.



The *ANSYS Dynamic Analysis Options* dialog controls the master degrees of freedom that will be used in your modal analysis, as well as the number of modes or frequencies that will be calculated. It includes the following sections/fields:

Translate Master DOF Set

If you have defined the master degrees of freedom that you want as a separate constraint set, check *Translate Constraint Set*, and choose the set from the drop-down list. If not, you may need to check the *Automatic DOF Selection Enable* option.

Automatic DOF Selection

Enable instructs ANSYS to automatically select the master degrees of freedom. Enter the *Number of DOF* that you want ANSYS. If you only want ANSYS to select translational degrees of freedom, check the *Exclude Rotational DOF* option.

Modal Expansion

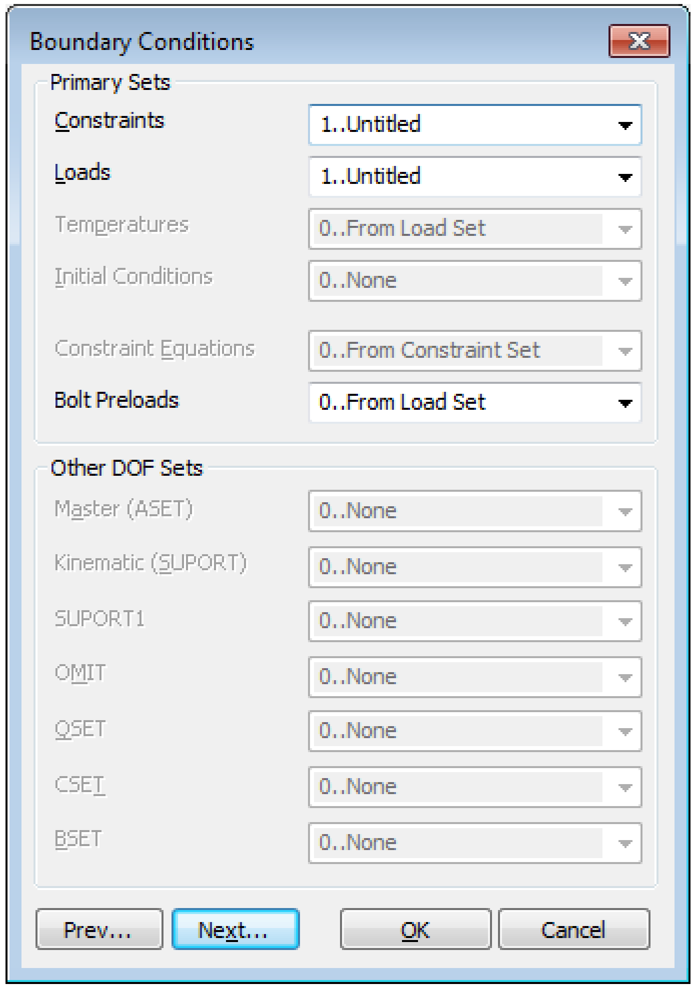
For Rev 5, you can:

- select the *Number of Modes* to extract
- limit modal extraction to a *Min Frequency/Max Frequency* range
- calculate elemental results (forces, stresses...) with the *Calculate Element Results* option; otherwise, only nodal results (mode shapes) will be computed

These options only apply if you are writing solution commands to the file.

Boundary Conditions

The *Boundary Conditions* dialog box lets you select the loads and constraints to apply to your analysis. You can apply boundary conditions as both master boundary conditions or in cases (depending on analysis type). Once you have entered this data, pick *Next* to continue setting up the analysis.

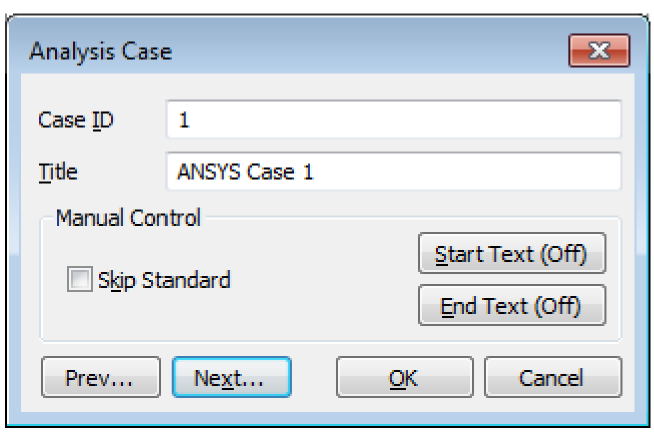


On this dialog box, the fields available for ANSYS are:

- *Constraints*: pick a constraint set for your model.
- *Loads*: pick a load set for your model.

Cases

Cases let you perform multiple analyses with different combinations of load and/or constraint sets. The analysis program will generate one output set for each case.



On this dialog box, you can enter a *Case ID, Title* and *Manual Control* options. Once you have entered this data, pick *Next* to continue defining the case. (The master requests and conditions provide the defaults for the cases.

The *Manual Control* options include:

- *Skip Standard*: If this switch is on, the interface will not write the load, constraint, and load option data to the ANSYS input file. *Start* and *End Text* will be written to the input file.
- *Start Text*: Pick this option to add text to the beginning of the load, constraint, and load option data in the input file.
- *End Text*: Pick this option to add text to the end of the load, constraint, and load option data in the input file.

8.3.2 Writing an ANSYS Model with File, Export

Note: This method is obsolete and has been removed from the default configuration of FEMAP. If you need to use it for any reason, you must use the *File, Preferences* command, click the *Interfaces* tab, and check the option for "Enable Old Analysis Interfaces". The use of this method for translation is NOT recommended. Use the *Model, Analysis* method instead.

The *File, Export, Analysis Model* command lets you write your FEMAP model into a file that can be read by ANSYS PREP7. This file contains PREP7 commands, just like the ones you would use if you were going to run ANSYS directly.

Starting to Export

After you choose *File, Export, Analysis Model* and select ANSYS, there are ten write options available: Static, Modes/Eigenvalues, Transient Dynamic/Time History, Frequency/Harmonic Response, Random Response, Buckling, Nonlinear Static, Nonlinear Transient Response, Steady State Heat Transfer, and Transient Heat Transfer.

Pick the type of analysis that you plan to perform. Next, you will see the *ANSYS Model Write* dialog box.

ANSYS Model Write

This dialog box includes the following fields:

Revision

It is very important that you set this option correctly for the version of ANSYS that you will be using. There are significant changes in commands and conventions between ANSYS Revisions 4.4 and 5.x And Above. If you choose the incorrect version, your model will almost certainly fail to run. Revision 5.x And Above is the default. Furthermore, only static and modal analysis are available for ANSYS Revision 4.4.

Title

Enter a title for the analysis.

Skip Preprocessor Command

If you check this option, FEMAP will not write the /PREP7 line at the top of the file.

Write All Groups as Components

If you check this option, FEMAP will write all groups as ANSYS "Components".

Write Commands for Automatic Solve

If you check this option, FEMAP will write additional commands to the end of your ANSYS file. These commands automatically perform the analysis when you load the file into ANSYS or run the model in batch mode.

Do not select this option if you are running a static, modal, or nonlinear static analysis and want to load the model into ANSYS, and then review it in PREP7 before beginning your analysis.

Use Iterative Solver

This option invokes the ANSYS iterative solver, and is only applicable for ANSYS v5.3+.

Large Deformation Effects

For nonlinear analysis (both static and transient), you also have the option to calculate *Large Deformation Effects*.

Analysis Memory

Enter the amount of memory to allocate for the analysis. If you leave the default of 0, ANSYS will automatically calculate the memory needed.

Specifying the File

When you press *OK*, you will see the standard file access dialog box. You must specify the name of the file that you want to create. The default file extension will be ".ANS", but you can choose any file name.

Preparing for Static and Nonlinear Static Analysis

After you choose a file, FEMAP will write your model geometry to the file, then ask for the constraints and loads that you want to translate. A series of dialog boxes will be displayed.

1. Pick the constraint set (*Select Constraint Set to Translate* dialog box).
2. Choose one or more load sets to go with that constraint set (*Entity Selection* dialog box).
3. Create additional constraint and load sets as desired.

With these dialog boxes, you can combine multiple FEMAP constraint and load sets into any number of ANSYS loading conditions.

You may also define multiple load and constraint sets for nonlinear static analysis; however, the first load set chosen must have active nonlinear analysis options. Pick *Model, Load, Nonlinear* to set the *Load Set Options for Nonlinear Analysis*. For details, see Section 4.3.5.1, "Model, Load, Nonlinear Analysis..." in *FEMAP Commands*.)

If nonlinear analysis options are not defined, FEMAP cannot translate the load sets. If nonlinear analysis options are defined for the first load set, these options will be used for all following load sets since ANSYS does not allow modifications after entering the solution process.

Preparing for Modal Analysis

If you chose a modal analysis type, you will see the *ANSYS Dynamic Analysis Options* dialog box.

The *ANSYS Dynamic Analysis Options* dialog controls the master degrees of freedom that will be used in your modal analysis, as well as the number of modes or frequencies that will be calculated. It includes the following sections/fields:

Translate Master DOF Set

If you have defined the master degrees of freedom that you want as a separate constraint set, check *Translate Constraint Set*, and choose the set from the drop-down list. If not, you may need to check the *Automatic DOF Selection Enable* option.

Automatic DOF Selection

Enable instructs ANSYS to automatically select the master degrees of freedom. Enter the *Number of DOF* that you want ANSYS. If you only want ANSYS to select translational degrees of freedom, check the *Exclude Rotational DOF* option.

Modal Expansion

For Rev 5, you can:

- select the *Number of Modes* to extract
- limit modal extraction to a *Min Frequency/Max Frequency* range
- calculate elemental results (forces, stresses...) with the *Calculate Element Results* option; otherwise, only nodal results (mode shapes) will be computed

These options only apply if you are writing solution commands to the file.

Preparing for Transient and Nonlinear Transient Analyses

Preparing for these types of analyses is very similar to preparing for static and modal analysis except that you may select only one load set and one constraint set.

To prevent translation errors, be careful to set appropriate options for both the dynamic load set (set from *Model, Load, Dynamic Analysis*) and the analysis type (set from *File, Export, Analysis Model*). The interface won't write the load set to the ANSYS solver if the *Solution Method* is turned off. It also won't write the load set if the *Solution Method* (for example, *Modal Frequency*) doesn't match the analysis type (for example, Transient Dynamic/Time History).

If the analysis type and the solution method for the load set) match, FEMAP will automatically determine whether to use a FULL (direct) or MODAL analysis method. If a direct method is active in the *Dynamic Analysis Solution*

Option, the *Constraint Set* dialog box appears and you can select the appropriate constraint set, after which the translation is completed. If the modal analysis method is active, you will be able to define additional parameters *ANSYS Dynamic Analysis Options* dialog box.

Translating for nonlinear transient analysis is identical to transient analysis, except that the modal transient method is not available, nonlinear analysis options must be active, and you can choose to activate *Large Deformation Effects*. If a modal transient solution method is selected, or nonlinear analysis options are not active, an error message appears and the load set is not written.

Preparing for Frequency/Harmonic Response Analysis

Preparing for frequency analysis is identical to transient analysis except for an additional dialog box which appears at the end of translation. This dialog box is identical to the *ANSYS Dynamic Analysis Options* dialog box and is used to select the number of frequencies and the frequency range to be analyzed. If a modal frequency solution method is chosen, this dialog box will appear twice. The first time you must choose the options for the initial modal analysis, while the second time it appears you can only choose the *Number of Frequencies, Min/Max Frequency,* and whether to calculate element results. All other options are disabled.

Preparing for Random Response Analysis

Preparing for random analysis is identical to transient dynamic/time history analysis, except that:

- the *ANSYS Dynamic Analysis Options* dialog box always appears because a modal analysis is required before every random analysis.
- you can select multiple load sets. All displacement values in the chosen load sets will be exported as base excitations (D), while all forces will be applied as nodal excitations (F).
- you can define output for a random analysis with the *ANSYS Output for Random Analysis* dialog box.

Preparing for Buckling Analysis

Buckling analysis simply requires the writing of a Static Solution followed by a Buckling Solution. Only one load set and one constraint set may be chosen for Buckling Analysis. The only other input required is the number of modes to be extracted and whether Elemental Results should be calculated. These are selected in the standard ANSYS Dynamic Analysis Options dialog box.

To prepare for a buckling analysis:

1. Build and mesh the FEMAP model. Apply one constraint set and one load set.
2. When you export the model, use the *ANSYS Dynamic Analysis Options* dialog box to enter the number of modes to be extracted and whether elemental results should be calculated.

Notes on ANSYS Dynamic Analyses

This topic reviews the limitations on transient and frequency response loading conditions in ANSYS. Loads on the same node or elemental face must have the same time history (transient) or same phase (frequency). If they do not, FEMAP will write the loading conditions assuming the last time history/phase, and you will most likely get results for a different loading condition than desired.

Also, with regard to Rayleigh damping, you can define alpha and beta damping for frequency analysis by defining the overall structural damping coefficient (G) and in some cases, the frequencies for element and system damping.

For frequency and random response analyses, FEMAP will automatically compute alpha and beta. If only one frequency is defined for the analysis, FEMAP assumes alpha is 0, and computes beta from G/wi where wi is the frequency. If a range of frequencies are specified, FEMAP will compute alpha and beta based on the equation

```
G = alpha/wi + beta*wi
```

by assuming G is constant over the frequency range. Two simultaneous equations are produced at the two frequencies which define the range.

For Rayleigh damping in transient analysis, FEMAP uses the *Frequency for System Damping (W3 - HZ)* and *Frequency for Element Damping (W - Hz)* input on the *Load Set Options for Dynamic Analysis* dialog box (*Model, Load, Dynamic Analysis)* to compute alpha and beta damping values. Alpha is simply the product of the overall structural damping coefficient (G) and W3 (*Frequency for System Damping*). Beta is G divided by W4 (*Frequency for Element Damping*). If G is zero, Rayleigh damping is ignored. Also, if W3 or W4 is zero, alpha or beta, respectively, will not be written.

Preparing for Steady-State Heat Transfer Analysis

Preparing for steady-state heat transfer analysis is just like preparing for static analysis, except that instead of defining nodal constraints, you define nodal temperatures (in a load set) for your boundary conditions.

Preparing for Nonlinear Contact Analysis

Surface-to-surface contact is defined in ANSYS using a combination of contact surface and target segment elements. ANSYS looks for contact only between contact surfaces and target segments that share the same real constant set. To define contact for ANSYS:

- Define the real constant set using the *Model, Contact, Contact Property* command. Pick the *ANSYS...* button to modify the KEYOPTs.
- Define the target segments using the *Model, Contact, Segment Surface* command. The target segments (TARGE169 (2-D) and TARGE170 (3-D)) can be either rigid or deformable. When defining a rigid target, you can use only line elements or Face 1 of a plate element.
- Define the contact surfaces using the *Model, Contact, Segment Surface* command. The contact surfaces can be 2-D (CONTA171 and CONTA172) or 3-D (CONTA173 and CONTA174), and must be deformable.
- Model a contact pair using the *Model, Contact, Contact Pair* command. The dialog box asks for a *Master* (the target segment) and a *Slave* (the contact surface), as well as a *Property* (the shared real constants set).
- Set up the time steps for a nonlinear analysis using the *Model, Load, Nonlinear Analysis* command.

For more information about these commands, see Section 4.4, "Creating Connections and Regions"in *FEMAP Commands*.

Hint: To create a rigid entity, use line elements for 2-D analysis and plate elements with the element normal direction facing the contact for 3-D analysis.

Special Cases

Depending on the contents of your model, you may see additional questions or warnings as the file is translated. For example, ANSYS requires certain IDs, especially for coordinate systems. If your model has entities that are outside of the allowable ID range, FEMAP will ask if you want to renumber prior to translating. Likewise, axisymmetric and other 2-D elements must lie in the global XY plane for ANSYS. If you have built your model in a different global plane, FEMAP will ask if you want to automatically realign it to the XY plane. FEMAP can only “flip” between global planes. You should never build an axisymmetric model in a skewed plane.

For axisymmetric models, you will also have an opportunity to specify a scale factor for loads. This factor is normally 1.0, but can be adjusted depending on whether you specified loads on a per radian or per revolution (360 degrees) basis. This is especially important since the required conventions change between ANSYS Revision 4.4 and 5.0.

8.3.3 Performing an ANSYS Analysis

Depending on where you plan to run ANSYS, the preliminary steps can be somewhat different. For example, if you are not running ANSYS on the same computer as FEMAP, you will have to transfer the file to the computer where ANSYS resides. Once that has been done, however, there are three basic approaches to running the file that you just created:

- From the Analysis Set Manager, pick the *Analyze* button. This requires you to have ANSYS on the same computer as FEMAP, and it must be properly setup. Before you start FEMAP, you must first define an environment variable in Windows. Name the variable ANSYS_EXE, give it a value equal to the complete file/path name of the ANSYS executable. For example:

```
ANSYS_EXE = c:\ansys\AnsysXX.exe
```

Where “XX” is the version of ANSYS (`i.e. c:\ansys\Ansys90.exe`) Once this variable is set, FEMAP will automatically launch ANSYS when you press *Analyze* from the *Analysis Set Manager*, or once the input file is written from *File, Export*. FEMAP will monitor the analysis and return results when it is finished.

It is sometime necessary to set the default ANSYS product so ANSYS uses the appropriately licensed product when running in batch mode.

To define the startup product for ANSYS add the following environment variable:

```
ANSYS90_PRODUCT=product_type
```

where product_type is the variable defining the product for which you are licensed (i.e. set ANSYS90_PRODUCT=ANE3FL)

for more on setting the ANSYS product see “Changing the Default Product for Start-up on Windows” in the ANSYS documentation.

NOTE: When changing environment variables you must close and restart FEMAP before the environment changes take effect.

- To run interactively, start ANSYS, and enter the command:

  ```
  /INPUT, filename, ANS
  ```

  where “filename” is the name of the file that you created. This assumes that the file is in the current directory and you used the default filename extension.

- To run in batch mode, you can specify the name of your model on the ANSYS command line, as follows:

  ```
  ANSYS -I filename.ANS
  ```

  Use the -O command line option to name the output file, and the -J option to name other files. If you are planning to run in batch mode, make certain you have FEMAP write the commands for automatic solution in your file.

8.3.4 Reading ANSYS Models

Just as you can translate a FEMAP model to ANSYS, you can also read ANSYS models into FEMAP. Since FEMAP reads PREP7 commands, there are several possible files that can be read.

If you built your model with PREP7, you should have an ANSYS log file that contains a list of all of your commands. If you only used commands that FEMAP supports, you can read your log file. In most cases however, this will not be possible. The log file will probably contain meshing or other model generation commands that cannot be read. In that case, load your model into ANSYS and use the CDWRIT command to produce an expanded file (normally FILE28) as follows:

```
CDWRITE,,,,,,,UNBLOCKED
```

The CDWRIT command expands meshing and other generation commands into a format that FEMAP can read. Reading an expanded file does not guarantee that every command will be read, but it certainly increases the number of commands that are supported.

When you begin to read a file, you will be asked for the ANSYS Revision level (4.4 or 5.0 - 9.0) of the file that you are going to read. Again, just like when you write a file for ANSYS, this is an important selection. Due to the differences in conventions, it is very unlikely that your model will translate correctly if you choose the wrong revision.

The only other input required is to select the file that you want to read using the standard file access dialog box.

8.3.5 Reading ANSYS Analysis Results

When you have completed your ANSYS analysis, you can load the results into FEMAP for post-processing. You must always start with a FEMAP model that corresponds to the ANSYS analysis. If you do not have one, or if you have changed the ANSYS files since you translated them from FEMAP, you must first start a new FEMAP model and read the ANSYS model, as described in the previous section.

Revision 4.4

For Rev 4.4 (and before), FEMAP can read two types of ANSYS output files: the binary output found in the standard ANSYS FILE12, or the “formatted” output produced by the BCDCNV command and typically written to FILE14. In either case, FEMAP assumes a default file extension of “.DAT”. You do not have to tell FEMAP which type of file you are reading. You simply specify the file name and FEMAP determines the type based on the contents of the file.

If you are running ANSYS on the same computer as FEMAP, the binary FILE12 is usually the best approach. If you are running FEMAP on a different computer, or are having trouble reading the binary files, the formatted files can often provide a more reliable alternative.

Revision 5.0 - 9.0

FEMAP reads the standard binary results file from Rev 5 to Rev 9.0. This file is created automatically by ANSYS, and FEMAP assumes a default file extension of “.RST” for structural results and “.RTH” for thermal results. The results file is in ANSYS external file format and can be moved between computers so that FEMAP can read it. These files are binary and should be transferred between computers as binary files.

To load your results, choose the *File, Import Analysis Results* command and select *ANSYS*. After you select the ANSYS revision, FEMAP will display the standard file access dialog box so you can choose the file that you want to read. Before reading data from a file, you will see a brief description of the file in the Messages window, and you will be asked to confirm that this is the file that you want to read.

Revision 10.0 and Above

FEMAP reads the standard binary results file from 10.0 and Above. Similar to 5.0 - 9.0, this file is created automatically by ANSYS, and FEMAP assumes a default file extension of “.RST” for structural results and “.RTH” for thermal results. The results file is in ANSYS external file format and can be moved between computers so that FEMAP can read it. These files are binary and should be transferred between computers as binary files.

Output for Post-processing

The format and contents of the ANSYS output files are described in the ANSYS User’s Manual, Programmers Manual, and in the description of the ANSYS Element Library.

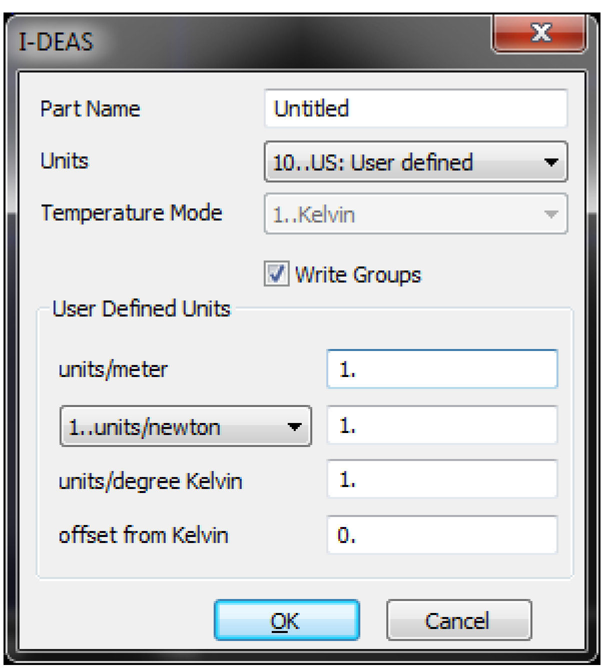
You must have a FEMAP model before reading the output file. FEMAP does not read the model information from your results file. FEMAP does read the displacements, elemental forces, stresses and strains, and reaction forces.

8.4 I-DEAS Interfaces

There are two direct interfaces between FEMAP and I-DEAS. You can either write a FEMAP model to the I-DEAS universal file format, or read an existing I-DEAS model. Both of these interfaces use the I-DEAS Simulation universal file datasets. These datasets may not be compatible with all versions of I-DEAS, or with universal files created in other applications. You cannot currently transfer geometry (points, curves, surfaces, or volumes) between I-DEAS and FEMAP using these interfaces. I-DEAS geometry, however, can be read into FEMAP using the Interoperability Data Interface (IDI) file format.

For more information on the entities that are translated, see Section 7.1, "Translation Table for ANSYS, I-DEAS, NASTRAN, and MSC Patran".

8.4.1 Writing an I-DEAS Model



To translate your model to I-DEAS universal file format, choose the *File, Export, Analysis Model* command, select the I-DEAS format, and press *Write*. You will see the standard file access dialog box, with the default extension set to “.UNV”. Select the file name that you want, and press *OK*. You will then see the I-DEAS dialog box, which enables you to control the part name and unit system that I-DEAS will use to interpret the universal file data.

Selection of part name and unit system is very important if you have coordinate systems or material data in your model.

Part Name:

Here you can specify the name of the part in I-DEAS to contain your coordinate systems. You can specify any name you want, up to 40 characters.

Units:

Since the I-DEAS universal file is written in terms of a specific unit system, you must choose the same unit system that you used to define your FEMAP model. FEMAP will not convert your model during the translation, with the following exceptions:

- Coordinate system locations are always written in meters in I-DEAS universal file datasets.
- Material data is converted to meter newton (SI) if the unit system is set to *User Defined*.

All I-DEAS unit systems are available, including *User Defined*. If you select *User Defined*, you can define scale factors for length, mass (or force) and temperature, and also define temperature offset.

Temperature Mode:

The temperature mode controls how I-DEAS interprets FEMAP temperature data. If the system selected is metric, the choice is Celsius or Kelvin. Otherwise the choice is Fahrenheit or Rankine.

Write Groups:

Checking this option will write FEMAP groups out to the I-DEAS universal file in the proper format.

User Defined Units:

These controls let you:

- enter a scale factor for length
- select whether to define a mass or force scaling factor
- enter a mass or force scaling factor
- enter a temperature scaling factor
- enter a temperature offset

I-DEAS will use these values to interpret the data in the universal file.

After you have selected any appropriate options, FEMAP will immediately write the I-DEAS universal file. You can read this file into I-DEAS using the *File, Import* command.

8.4.2 Reading an I-DEAS Model

Reading an I-DEAS Simulation universal file is even easier than writing one. Open a new model, choose the *File, Import, Analysis Model* command, and select the I-DEAS format. You will see the standard file access dialog box, and you can choose the file that you want to read. There are no options when reading an I-DEAS file. FEMAP simply reads all of the universal file datasets that it can interpret.

Coordinate system data is always written to universal files in meters. Material data is always written in the units it was created in (not necessarily the same as the units that the rest of the universal file datasets are written in). Both coordinate system and material data are converted when read into FEMAP.

If you wish to work in FEMAP in units other than those defined in the universal file, several FEMAP units conversion factor files are available to convert between I-DEAS unit systems. Use the *Tools, Convert Units* command to apply these to the FEMAP model.

8.5 LS-DYNA Interfaces

FEMAP provides direct interfaces to the LS-DYNA file formats. You can write a FEMAP model for analysis in LS-DYNA, or read analysis results for post-processing.

Topics in this section include:

- Section 8.5.1, "Writing an LS-DYNA Model with Model, Analysis"
- Section 8.5.2, "Writing an LS-DYNA Model with File, Export" (Obsolete)
- Section 8.5.3, "Performing an LS-DYNA Analysis"
- Section 8.5.4, "Reading an LS-DYNA Analysis Model"
- Section 8.5.5, "Post-processing LS-DYNA Results"

For more information on the entities that are translated, see:

- Section 7.2, "Translation Table for ABAQUS, LS-DYNA, and MSC.Marc"

8.5.1 Writing an LS-DYNA Model with Model, Analysis

The *Model, Analysis* command opens the *Analysis Set Manager*, which enables you to write a FEMAP model to LS-DYNA. Alternatively, you can also use the *File, Export, Analysis Model* command (see Section 8.5.2, "Writing an LS-DYNA Model with File, Export") but this is not recommended.

8.5.1.1 General Analysis Process Overview

The *Analysis Set Manager* lets you create an analysis set, which is an input file for LS-DYNA. To define an analysis set:

1. Pick *Model, Analyze* to open the *Analysis Set Manager* dialog box.
2. Pick the first item on the list, then pick *New*. (You can also double-click on the item).
3. On the *Analysis Set* dialog box, choose *28..LS-DYNA* as the *Analysis Program*. Select the *Analysis Type: 9..Explicit Transient Dynamics* or *13..Implicit Transient Dynamics*. Enter a title. The title is simply exported as the content of the *TITLE command in the LS-DYNA input file.

   You can also choose to use a *Linked Solver* or *VisQ*. See Section 2.6.2.6, "Solvers" or Section 4.10.2.1, "Run Analysis Using Linked Solver / VisQ / Local Settings" in the FEMAP Commands Manual for more information.
4. When you choose *13..Implicit Transient Dynamics*, FEMAP will write out the following entries by default:

   *CONTROL_TERMINATION with a 0. in the first field

   *CONTROL_IMPLICIT_TERMINATION with 0.0 in the first 5 fields and a 3 in the 6th field

   *CONTROL_IMPLICIT_GENERAL with a 1 in the 1st field and a 0. in the 2nd field

   *CONTROL_IMPLICIT_AUTO with 1 in the 1st field, nothing in the 2nd and 3rd fields, and 0.01 in the 4th field

   *CONTROL_IMPLICIT_SOLVER with a 0 in the first 3 fields

   *CONTROL_IMPLICIT_SOLUTION with 0 in fields 1, 2, 3, 5, 6, and 7 on the first line and the first 4 fields on the second line along with a 0.0 in the 4th field of the first line
5. Pick *Next* to open the *LS-DYNA Model Options* dialog box. (Alternatively, you can pick *OK* to close the *Analysis Set* dialog box. From the analysis set list, you can then double-click on an option to bring up the dialog box where the option is defined.) Select the *Export Options* for this analysis.
6. Pick *Next* to open the *Master Requests and Conditions* dialog box. The “master requests and conditions” are the default boundary conditions. Enter a title.
7. Pick *Next* to open the *LS-DYNA Analysis Control* dialog box. Select the *Analysis Conditions*, *Analysis Info*, *Solver Options*, and *Additional Output* requests for this analysis
8. Pick *Next* to open the *Boundary Conditions* dialog box. Select the constraints and loads.

See Section 8.5.1.2, "Setting the LS-DYNA Analysis Parameters" for descriptions of the LS-DYNA interface dialog boxes

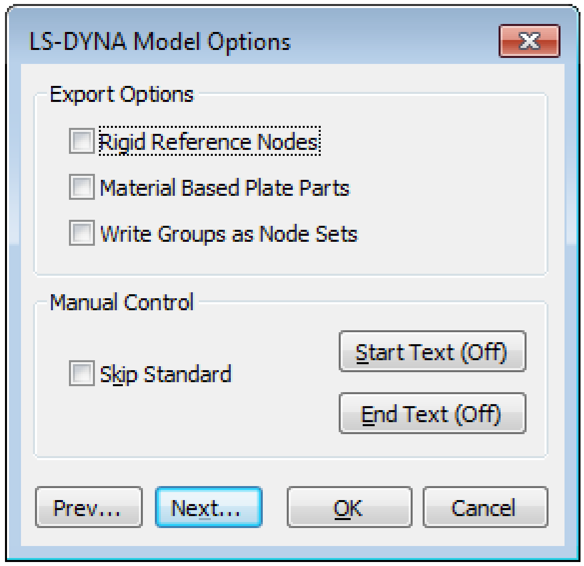
8.5.1.2 Setting the LS-DYNA Analysis Parameters

To set up an LS-DYNA analysis, you will work through the following dialog boxes:

- "LS-DYNA Model Options"
- "Master Requests and Conditions"
- "LS-DYNA Analysis Control"
- "Boundary Conditions"

LS-DYNA Model Options

The *LS-DYNA Model Options* dialog box contains *Export Options* and a *Manual Control* section to enter text into the input file using the *Analysis Set Manager*.



Export Options

This section controls export options for rigid body reference nodes and plate parts.

Rigid Reference Nodes

Rigid references nodes are used in FEMAP to define motion of a given rigid body, instead of requiring prescribed conditions on all nodes of the rigid body. Many times this node will be a node in the model that is free (massless) whose sole purpose is to describe the motion of the rigid body. By default, FEMAP will find these nodes and prevent them from being written to the LS-DYNA input file. The motion of the node, as well as any forces or moments, will then be assigned to the rigid contact segment that references the node.

There may be times, however, when you would like to export this node. By turning on this option, FEMAP will export these nodes just like any other nodes in the model.

Note: Prescribed motion and forces are only assigned to the rigid body if you use a material *Other Type 20*, LS-DYNA rigid material for the property associated with the contact segment

Material Based Plate Parts

This option will cause FEMAP to write all parts based upon the materials that they reference, not the properties. It should be typically left off unless you have a variable thickness shell model. This option is very useful in this circumstance.

Since FEMAP defines the plate thickness on the property card, there is one property for every thickness variation. FEMAP also typically exports each part based upon the property. If you have a large variable thickness plate model, there could be literally thousands of PARTs written by FEMAP, possibly accessing the same material. When this option is checked, however, FEMAP will build the parts based on the common materials, and the output will contain only as many parts as there are materials. Be careful, however, to define different materials for each part, even if they are the same material, because without reference to the property, FEMAP will have no way of telling the difference between parts, except by material ID.

Write Groups as Node Sets

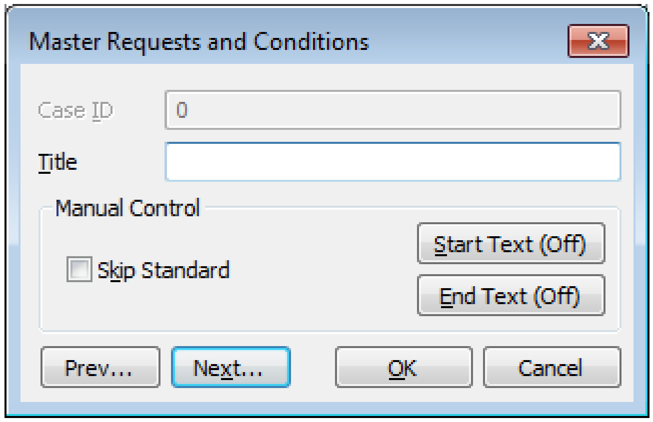
Writes all FEMAP groups as Node Sets in the LS-DYNA input file. This option is checked ON by default.

Manual Control:

- *Skip Standard*: If this switch is on, the interface will not write the load, constraint, and load option data to the LS-DYNA input file. *Start* and *End Text* will be written to the input file.
- *Start Text*: Pick this option to add text to the beginning of the Export Options in the input file.
- *End Text*: Pick this option to add text to the end of the Export Options in the input file.

Master Requests and Conditions

In an LS-DYNA analysis, the master requests and conditions are the default boundary conditions. The analysis will generate one output set for the master requests and conditions.



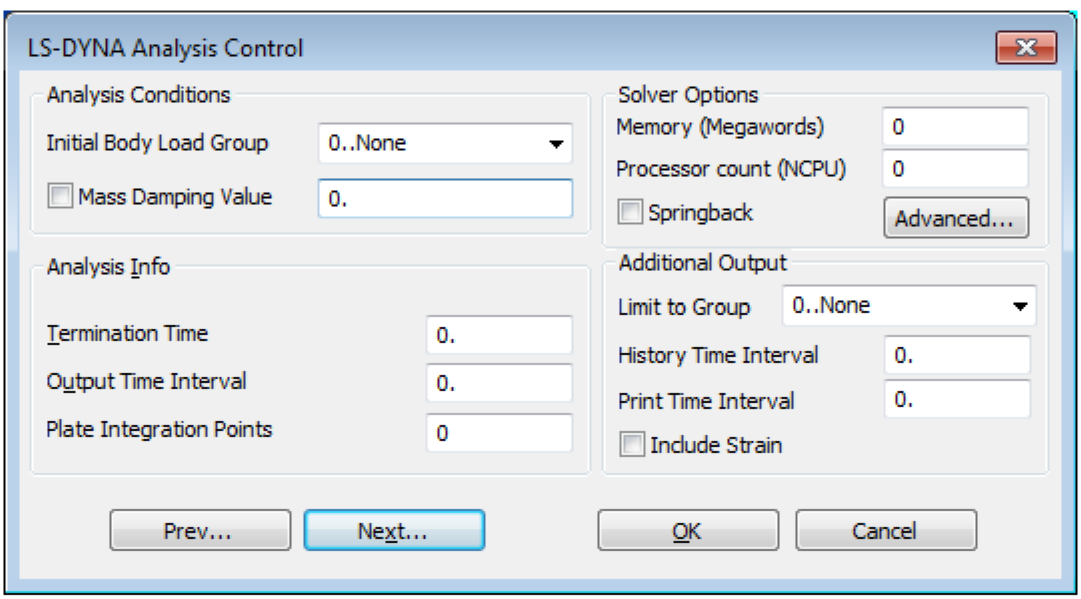
On the *Master Requests and Conditions* dialog box, you can enter a *Title* and *Manual Control* options. Once you have entered this data, pick *Next* to set up *LS-DYNA Analysis Control*.

The *Manual Control* options include:

- *Skip Standard*: If this switch is on, the interface will not write the load, constraint, and load option data to the LS-DYNA input file. *Start* and *End Text* will be written to the input file.
- *Start Text*: Pick this option to add text to the beginning of the load, constraint, and load option data in the input file.
- *End Text*: Pick this option to add text to the end of the load, constraint, and load option data in the input file.

LS-DYNA Analysis Control

Analysis Conditions



Initial Body Load Group

When you have an angular velocity defined as a body load in FEMAP *(Model, Load, Body)* this option allows you to select a single Group to apply the angular velocity. The Group MUST contain a property.

If you have a Group with a property and an angular velocity body load in the load set designated for *Initial Conditions* in the *Boundary Conditions* dialog box, then the *INITIAL_VELOCITY_GENERATION card will be written out for this property only instead of the whole model.

Mass Damping Value

If the *Mass Damping* option is on, FEMAP will export the value you enter to the *DAMPING_GLOBAL card. This will also override any material-based mass damping associated with the model.

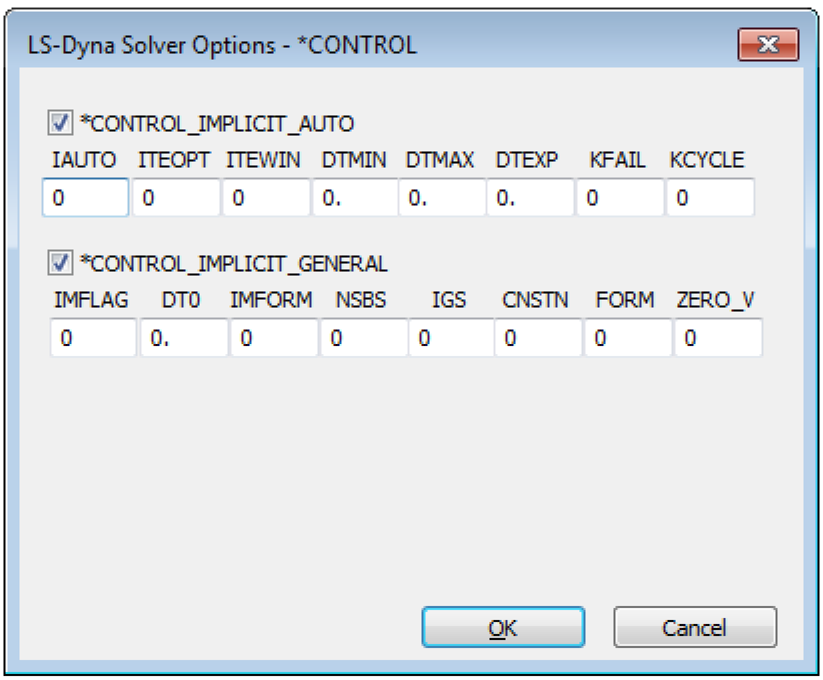
If a mass damping value is not entered, damping will not be considered in the model unless the you specify either the *Frequency for System Damping* (W3) or the *Frequency for Element Damping* (W4) under *Model, Load, Dynamic Analysis*, You must also define a damping value for the material. If these values are specified, FEMAP will export damping values for each part that has a damping value for the material. The mass damping value is simply the product of the damping value on the material card (2*C/Co) and W4. The results are exported to the *DAMPING_PART_MASS card. Stiffness damping is obtained by dividing the material damping value by W3. The result is then exported to the *DAMPING_PART_STIFFNESS card.

Entering a global mass damping value will prevent the export of the DAMPING_PART_MASS card. A*DAMPING_PART_STIFFNESS card, however, will be written for all materials with a damping value as long as W3 is not zero.

Solver Options

Checking *Springback* will write out the *CONTROL_IMPLICIT_SOLVER card with a "2" in the first field. If both are selected, the *Springback* option takes precedence. In addition, you can specify *Memory* in Megawords and the *Processor count* using an integer value.

The *Advanced...* button is only available when using *Analysis Type* is set to *13..Implicit Transient Dynamics*. When clicked, the *LS-Dyna Solver Options - *CONTROL* dialog box will appear.



This dialog can be used to enter values for the *CONTROL_IMPLICIT_AUTO entry and/or the *CONTROL_IMPLICIT_GENERAL entry. Consult the LS-DYNA Keyword User's Manual - Volume I for more information.

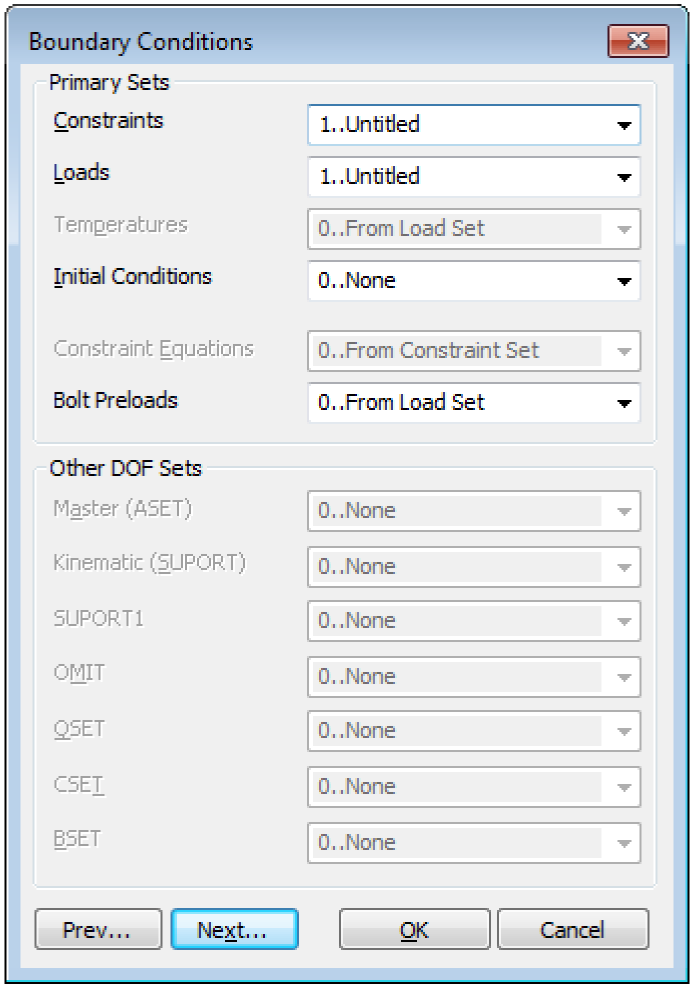
Analysis Info and Additional Output

These sections let you define *Termination Time, Output Time Interval,* and *Additional Output* information. The *Output Time Interval* is used to generate the D3PLOT post-processing file. Results for all nodes and elements are recovered at this time interval. Additional time intervals can be specified for time history (D3THDT), and ASCII (NODOUT and ELOUT) files. You may also limit output to these files to just nodes and elements contained in a FEMAP group. If no group is selected, results will be requested for all nodes and elements.

Note: A relatively large time step is typically chosen for the output interval in comparison to the ASCII and time history files. These outputs are typically used to obtain results at a very fine time step. Thus, output to these files are typically limited to a few key nodes and elements.

Boundary Conditions

Select the *Loads* and *Constraints* to use for this analysis, as well as the *Initial Conditions*. When you select a load set for *Initial Conditions*, only those loads which are pertinent to initial conditions, such as velocities and temperatures, will be exported from this load set. Since velocities and temperatures can also be applied throughout the history in the load set chosen under *Loads*, it is best to define a load set specifically for initial conditions.



On this dialog box, the fields available for LS-DYNA are:

- *Constraints*: pick a constraint set for your model.
- *Loads*: pick a load set for your model.
- *Initial Conditions*: pick a load set representing initial conditions for your model.

8.5.2 Writing an LS-DYNA Model with File, Export

Note: This method is obsolete and has been removed from the default configuration of FEMAP. If you need to use it for any reason, you must use the *File, Preferences* command, click the *Interfaces* tab, and check the option for “Enable Old Analysis Interfaces”. The use of this method for translation is NOT recommended. Use the *Model, Analysis* method instead.

The *File, Export, Analysis Model* command allows you to write your FEMAP model into a file that can be analyzed by LS-DYNA. Simply choose this command, select *LS-DYNA*, and *Explicit Transient Dynamics* under *Format*. Once you make these selections, you will see the standard file access dialog box, where you can specify the name of the file to be created. The default file name extension for this file is “.DYN”, but you can choose any name.

After choosing a file name, you will see the following *LS-DYNA Analysis Control* dialog box. This dialog box lets you specify a title for the analysis, the load and constraint sets, specific export options, and time step values.

The title is simply exported as the content of the *TITLE command in the LS-DYNA input file, while the remaining options are defined more fully below.

Analysis Conditions

Select the *Loads* and *Constraints* to use for this analysis, as well as the *Initial Conditions*. When you select a load set for *Initial Conditions*, only those loads which are pertinent to initial conditions, such as velocities and temperatures, will be exported from this load set. Since velocities and temperatures can also be applied throughout the history in the load set chosen under *Loads*, it is best to define a load set specifically for initial conditions.

Mass Damping Value

If the *Mass Damping* option is on, FEMAP will export the value you enter to the *DAMPING_GLOBAL card. This will also override any material-based mass damping associated with the model.

If a mass damping value is not entered, damping will not be considered in the model unless the you specify either the *Frequency for System Damping* (W3) or the *Frequency for Element Damping* (W4) under *Model, Load, Dynamic Analysis*, You must also define a damping value for the material. If these values are specified, FEMAP will export damping values for each part that has a damping value for the material. The mass damping value is simply the product of the damping value on the material card (2*C/Co) and W4. The results are exported to the *DAMPING_PART_MASS card. Stiffness damping is obtained by dividing the material damping value by W3. The result is then exported to the *DAMPING_PART_STIFFNESS card.

Entering a global mass damping value will prevent the export of the DAMPING_PART_MASS card. A*DAMPING_PART_STIFFNESS card, however, will be written for all materials with a damping value as long as W3 is not zero.

Export Options

This section controls export options for rigid body reference nodes and plate parts.

Rigid Reference Nodes

Rigid references nodes are used in FEMAP to define motion of a given rigid body, instead of requiring prescribed conditions on all nodes of the rigid body. Many times this node will be a node in the model that is free (massless) whose sole purpose is to describe the motion of the rigid body. By default, FEMAP will find these nodes and prevent them from being written to the LS-DYNA input file. The motion of the node, as well as any forces or moments, will then be assigned to the rigid contact segment that references the node.

There may be times, however, when you would like to export this node. By turning on this option, FEMAP will export these nodes just like any other nodes in the model.

Note: Prescribed motion and forces are only assigned to the rigid body if you use a material *Other Type 20*, LS-DYNA rigid material for the property associated with the contact segment

Material Based Plate Parts

This option will cause FEMAP to write all parts based upon the materials that they reference, not the properties. It should be typically left off unless you have a variable thickness shell model. This option is very useful in this circumstance.

Since FEMAP defines the plate thickness on the property card, there is one property for every thickness variation. FEMAP also typically exports each part based upon the property. If you have a large variable thickness plate model, there could be literally thousands of PARTs written by FEMAP, possibly accessing the same material. When this option is checked, however, FEMAP will build the parts based on the common materials, and the output will contain only as many parts as there are materials. Be careful, however, to define different materials for each part, even if they are the same material, because without reference to the property, FEMAP will have no way of telling the difference between parts, except by material ID.

Write Groups as Node Sets

Writes all FEMAP groups as Node Sets in the LS-DYNA input file. This option is checked ON by default.

Analysis Info and Additional Output

These sections let you define *Termination Time, Output Time Interval,* and additional output information. The *Output Time Interval* is used to generate the D3PLOT post-processing file. Results for all nodes and elements are recovered at this time interval. Additional time intervals can be specified for time history (D3THDT), and ASCII (NODOUT and ELOUT) files. You may also limit output to these files to just nodes and elements contained in a FEMAP group. If no group is selected, results will be requested for all nodes and elements.

Note: A relatively large time step is typically chosen for the output interval in comparison to the ASCII and time history files. These outputs are typically used to obtain results at a very fine time step. Thus, output to these files are typically limited to a few key nodes and elements.

8.5.3 Performing an LS-DYNA Analysis

FEMAP can launch LS-DYNA and automatically run your analysis, either from the File Analyze command or pressing Analyze in Model Analysis, if you first setup the analysis program. To do this, before running FEMAP, you must establish an environment variable named LSDYNA_EXE. Set the value of this environment variable to the full path to the LS-DYNA solver executable, for example:

```
SET LSDYNA_EXE = c:\lsdyna\ls960-nsmp.exe
```

Once this environment variable has been defined, FEMAP will be able to launch the analysis program and monitor the job until it is complete.

8.5.4 Reading an LS-DYNA Analysis Model

In addition to the ability to write LS-DYNA models and read in results, FEMAP can also read the nodal locations and element connectivity from the LS-DYNA binary post-processing files (D3PLOT and D3THDT). There are two ways to read this information. First, you can choose *File, Import, Analysis Model*, then choose the output file that you want to read. In addition, if you start with an empty model and select *File, Import, Analysis Results*, FEMAP can read both the model and results in one pass.

Note: FEMAP can currently only read nodes and element connectivity from the results file. The elements contain references to LS-DYNA properties and materials, so you can group the model by property or material, but actual records on the properties/materials will be all zero.

8.5.5 Post-processing LS-DYNA Results

When you have completed your LS-DYNA analysis, you can load results into FEMAP for post-processing. To load results, choose the *File, Import, Analysis Results* command and select *LS-DYNA*. FEMAP will display the standard file access dialog box so you can choose the file you want to post-process. FEMAP reads the binary D3PLOT and D3THDT files.

Both D3PLOT and D3THDT files are a series of files which must be imported. Simply select the D3PLOT or D3THDT file, and FEMAP will automatically read the entire series. If you have not used the default names for these files, you will need to change the *File of Type* to D*YNA State Database* or *DYNA Time History Database* to identify the format.

Once you select the file, you may see a question, “OK to read file as double precision format? The file does not seem to conform to single precision format.”. It is recommended that you answer *Yes* for this question when it appears. Next, the *DYNA Results* dialog box will appear. This dialog box allows you to specify the type of data that you want to recover, and the time steps to recover.

Data Recovery Options

There are three main sections under the type of output. Global variables such as kinetic energy can be read and stored in FEMAP. These values are stored as functions for plotting purposes. You can also specify the typical nodal and elemental results.

Step Recovery Options

This section allows you to limit the amount of output sets. You can specify the time range (*All, Before, After, Between, Final*), as well as every Nth output set and First Step in New File

Note: These options are very useful for quick check of an analysis to significantly reduce the import time and model size. You could quickly read in the last step, or only a few steps, check the contact conditions, and then either read in the full results for further post-processing or modify the model and perform another analysis.

Note: If you have beams in your model that have a vector defined for the orientation instead of a 3rd node, FEMAP automatically creates dummy nodes for the LS-DYNA input deck. These nodes do not exist in the FEMAP model, but results may be recovered for them. This will produce an error message when reading in LS-DYNA results. In this case, this error message (*Output contains entities that do not exist in your model*) can be ignored. The results should still be valid.

Time History Data

When importing time history data (D3THDT), you will also have the option to import all information as functions. You will typically want to keep this option on, since most history files contain large amounts of step data for a few nodes or elements. This type of output is handled much more efficiently as functions than as standard output. The FEMAP model size could be orders of magnitude larger when you do not save as functions for a time history file with a large number of steps for only a few entities.

Old Version Import

This option can be used when importing d3plot files from version before v970. If enable, it simply uses an older version of the import results translator.

8.6 Marc Interfaces

FEMAP provides direct interfaces to the MSC.Marc file formats. You can write a FEMAP model for analysis in MSC.Marc, or read analysis results for post-processing.

- Section 8.6.1, "Writing an MSC.Marc Model with Model, Analysis"
- Section 8.6.2, "Writing an MSC.Marc Model with File, Export" (Obsolete)
- Section 8.6.3, "Performing an MSC.Marc Analysis"
- Section 8.6.4, "Reading an MSC.Marc Analysis Model"
- Section 8.6.5, "Post-processing MSC.Marc Results"

For more information on the entities that are translated, see Section 7.2, "Translation Table for ABAQUS, LS-DYNA, and MSC.Marc".

8.6.1 Writing an MSC.Marc Model with Model, Analysis

The *Model, Analysis,* command allows you to write your FEMAP model into a file that can be analyzed by MSC.Marc 2001.

The export process can be separated into three areas: analysis parameters, model definition, and history definition.

- Parameters include options which set up sizing and initial switches for the analysis.
- Model definition include Model options, Contact Table, Boundary Conditions, and Output.
- History definition include History options, Contact Table, and Boundary Conditions.

The remainder of this section describes in detail the steps for defining a MARC analysis with *Model, Analysis*. You will:

1. Define an Analysis set. (See Section 8.6.1.1, "Analysis Set".)
2. Define the Solution Parameters. (See Section 8.6.1.2, "Analysis Parameters").
3. Define the Master case. (See Section 8.6.1.3, "Master Requests and Conditions").
4. Define Model Definition options. (See Section 8.6.1.4, "Model Definition")
5. Define a Contact Table. (See Section 8.6.1.5, "Contact Table".)
6. Define Boundary conditions. (See Section 8.6.1.6, "Boundary Conditions")
7. Define Output Requests. (See Section 8.6.1.7, "Output Requests")
8. Optionally, define the History Definition for additional cases that include different load and constraint sets. (See Section 4.10.1.6, "Cases"in *FEMAP Commands*. and Section 8.6.1.9, "MARC History Definition")

- History Definition will also include Load Increment options. (See Section 8.6.1.10, "Load Increment Options")

8.6.1.1 Analysis Set

When you create an analysis set, you define the solution parameters, model options, boundary conditions, output selection and history steps for the MSC.Marc analysis. An analysis set is stored with the model file, so you can reuse this data. You can also store an analysis set in a FEMAP analysis library.

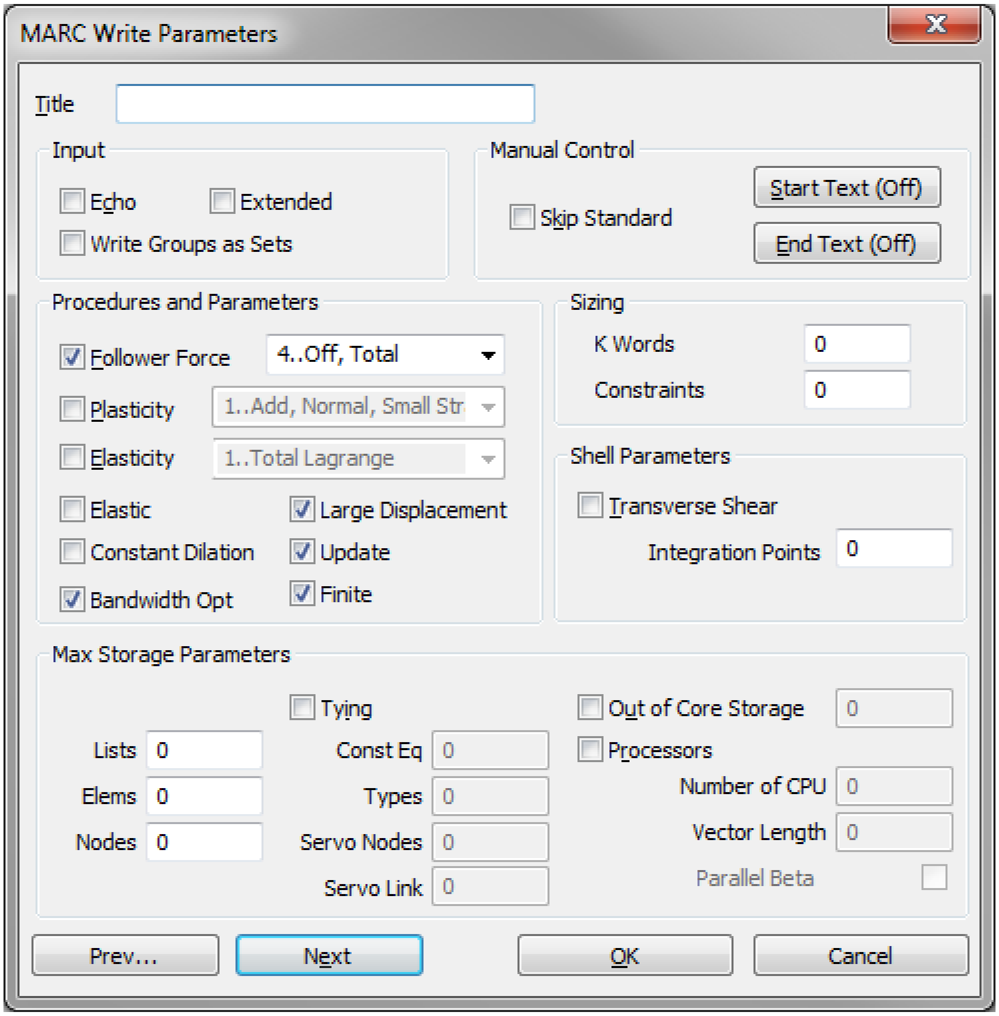
To define an analysis set:

1. Pick *Model, Analyze* to open the *Analysis Set Manager* dialog box.
2. Pick the first item on the list, then pick *New*. (You can also double-click on the item).
3. Choose the *Analysis Program* and *Analysis Type*. These options determine which element types (structural or heat transfer) will be written, and set the defaults for the other NASTRAN interface dialog boxes.

   You can also choose to use a *Linked Solver* or *VisQ*. See Section 2.6.2.6, "Solvers" or Section 4.10.2.1, "Run Analysis Using Linked Solver / VisQ / Local Settings" in the FEMAP Commands Manual for more information.This information determines the remaining options that you define.
4. The best way to define an analysis set is to pick *Next* to work through the dialog boxes in order. Alternatively, you can pick *OK* to close the dialog box. From the analysis set list, you can then doubleclick on a parameter to bring up the dialog box.

8.6.1.2 Analysis Parameters

After choosing a file name, you will see the *MARC Write Parameters* dialog box. This dialog box controls the writing of all parameters to MSC.Marc. The specific options are described below.



Title

Enter a title for the analysis. It can be written to both binary and ASCII output files.

Input

This section allows you to obtain an *Echo* of the input in the output (*.out) file, write the *Extended* parameter for extended input, and to *Write Groups as Sets*. The *Extended* parameter will be necessary if any IDs in the model are above five digits. Without this parameter, all nodal, elemental, property, and material IDs must be under 100,000 or the analysis will fail due to input errors.

You may also choose to write all FEMAP groups as sets. This is a convenient method to obtain sets which you can then easily apply loading or other conditions by directly manipulating the MSC.Marc input deck.

Manual Control

- If the *Skip Standard* switch is off, the software writes standard Parameter section.
- Pick *Start Text* to add text to the beginning of the Parameter section.
- Pick *End Text* to add text to the end of the Parameter section.

Sizing

This section determines the initial memory allocation (in thousands of Words - kWords) for the overall model, as well as constraints. The constraint value is not necessary unless you will have a History Definition in your model that contains more constraints than the Model Definition. If this section is left blank then MSC.Marc will determine the defaults.

Shell Parameters

This section determines whether the *Transverse Shear* option is chosen for a plate/shell analysis. You may also specify the number of integration layers through the thickness of plate elements. The default is 11 layers. Any input must be between 3 and 15, and be an odd number. If the model contains composites, this value is overridden by the number of composite layers.

Procedures and Parameters

This section enables you to select between many different parameters available in MSC.Marc.

Follower Force (Incremental Load)

This option controls whether follower forces are employed as well as whether loads applied on History Definitions are incremental or total. There are actually three different options involved with this one parameter: *Follower Force Stiffness* on or off, *Total* or *Incremental Loading*, and if *Follower Force* is on, (3) *Follower Force* is based upon *Displacement* at last iteration or beginning of increment. The one selection will set all three options.

Plasticity

This parameter controls the plasticity procedure that is used in MSC.Marc. There are currently five options for this parameter. They involve use of either the *Additive* or *Multiplicative Decomposition* method. If *Additive* is desired, you can also choose from two other options: *Mean Normal* or *Radial Return* method, and *Small or Large Strain* formulation. *Multiplicative Decomposition* requires radial return and large strain formulation. *Multiplicative Decomposition* is more accurate for large elastic and plastic strains, but it requires that all elasticity be isotropic. The default for this parameter if not set is *Additive Decomposition* using the mean normal method and small strain formulation.

Elasticity

This parameter controls the formulation for large strain elasticity. The default is a total Lagrange formulation. You may also select an updated Lagrange formulation. If you choose the total Lagrange formulation with Mooney or Ogden material models, the hybrid element formulation (Herrmann) must be used while the Foam material can use the standard elements. For updated Lagrange formulation, the standard elements amy be used for all three material types.

Elastic

This option can be implemented for elastic analysis with multiple loads. It essentially builds the stiffness matrix once, and then repeatedly back substitutes the load vectors to obtain the results. This option should not be used in nonlinear analysis, and will not be written if the *Large Displacement option* is checked.

Note: This option will also cause any constraint changes in the *History Definition* section to be ignored. Since the decomposition matrix is only formed once, at the end of the Model Definition stage (Increment 0), changes in constraints cannot be allowed.

Large Displacement, Update, Constant Dilation

These options invoke the *Large Displacement* option, the updated Lagrange procedure (instead of total), and *Constant Dilation*. Constant dilation is recommended for lower order elements in nearly incompressible analysis. This option can also be used for the individual elements by specifying a hybrid element formulation for the lower order standard element types (bricks, plane strain, and axisymmetric).

Bandwidth Optimization, Finite

Bandwidth Optimization uses bandwidth optimizers to reduce computer costs for larger problems. *Finite* uses the large strain plasticity option to include the effects of large inelastic deformation.

Max Storage Parameters

This section controls memory allocation for several different lists. Many of these parameters will only be required if a History Definition has a larger number of lists than the model definition.

Distributed Loads

This parameter controls the memory allocation for elemental and nodal loading. FEMAP will determine the number of entities and write the DIST LOADS option automatically. The user can override FEMAP by simply defining the numbers for Lists, Elems, and Nodes.

This is only required if you will be applying more loads in a History Definition than was originally on the Model Definition inputs include the total number of lists for distributed loads, the maximum number of elements in a *dist* *loads* list (pressure, distributed load, gravity), and the maximum number of nodes in a POINT LOAD list. FEMAP will export each load as its own list, therefore, the number of lists must be equal to or greater than the total number of loads. The number of nodes and elements in any specific list, however, will typically be 1 since each load has its own list containing one member. The one exception to this is FEMAP body loads (gravity and rotational velocity), which will contain the total number of elements in the model in one list.

TYING

This parameter allocates storage for tying data which includes FEMAP rigid elements (MARC Tying Constraints 1-6, 100) and constraint equations (MARC Servo Links).

The maximum number of MARC constraint equations should be set equal to the number of FEMAP rigid elements in the model, and the number of different types can be safely set to 7 (only 1-6 and 100 are currently supported). The *Servo Nodes* should be set to the maximum number of nodes in any constraint equation (set to 2 if no FEMAP constraint equations), and the *Servo Link* should be set to the maximum number of FEMAP constraint equations.

Out of Core Storage

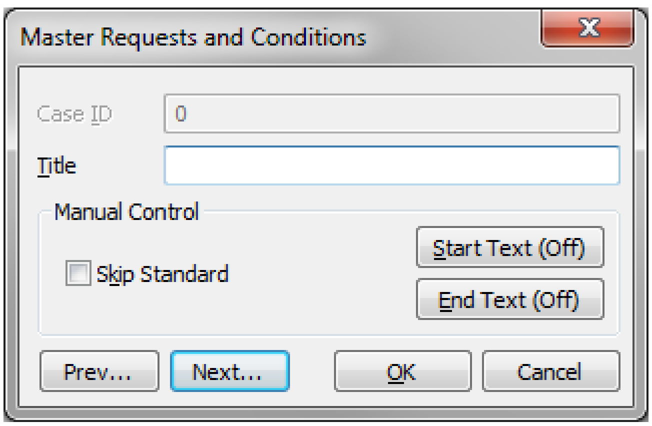
This option invokes the ELSTO parameter for large models. It will cause all elemental quantities to be stored in an auxiliary area (unit 3) to free core memory for calculations. The input for this option is in Words.

Processors

This option enables you to specify the number of CPUs, the vector length and whether the beta matrices are to be formed in parallel.

8.6.1.3 Master Requests and Conditions

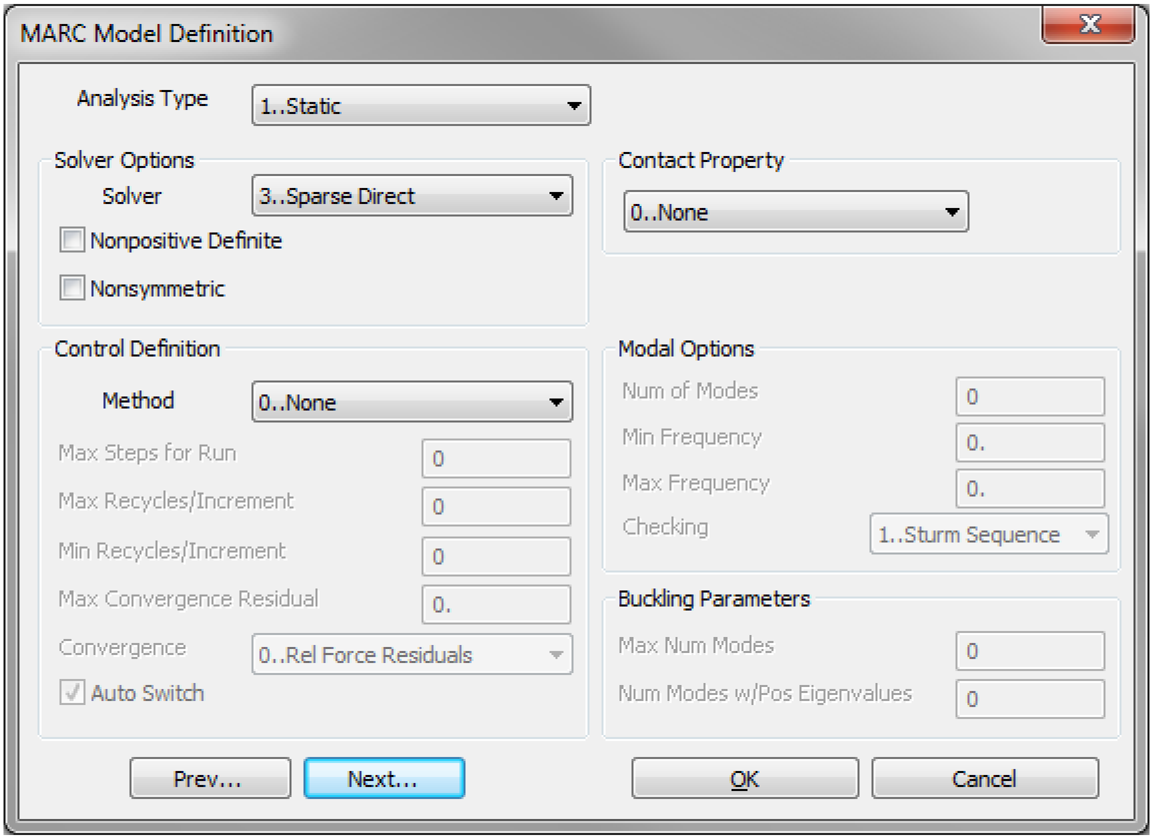
Master Requests and Conditions define the master output requests and boundary conditions (loads and constraints) for your analysis.



The software will write all of these requests and conditions to the Model section. In addition, you can define cases which are MSC.Marc history data.

8.6.1.4 Model Definition

Once you press *Next* on the *MARC Write Parameters* dialog box, the *MARC Model Definition* dialog box will appear.



This dialog box is separated into six major sections: *Analysis Type, Solver Options Control Definition, Contact Property, Modal Options, Buckling Parameters*.

Analysis Type

Pick the Analysis type to be used in the model definition, choose from 1) Static, 2) Normal Modes/Eigenvalue 3) Buckling.

Solver Options

This option allows you to chose from several different direct and iterative solvers. When this option is active, you can also select options for *Nonpositive Definite* and *Nonsysmmetric matrices*.

Control Definition

This section determines the type of solution method (including the convergence criteria), and the solver type.

Method

For the solution method choose from either *None, the Newton Raphson techniques (Full, Modified, or Strain Correction) or the Secant method*. If you choose a method, you can then also specify bounding parameters for the number of recycles, as well as the maximum steps for a run. If no values are entered, the defaults will be used.

You may also define the *convergence criteria*. You have six options in this box, which are really the combination of two outputs defining the basis of convergence: F*orce, Displacement,* or *Strain Energy* for either relative or absolute values. The selection of the *Nonpositive Definite* option in the Solver Options section will force solutions of problems with a nonpositive definite matrix.

Auto Switch

Enables the *Control* parameter for switching of convergence testing between residuals and displacements.

Contact Property

All contact segments in MARC are exported to the CONTACT option in MSC.Marc.

This option includes contact property information for all contact entities that are written. To define these values, simply create a contact property in FEMAP, and then select this property under *Contact Property*. If there are contact segments in the model, FEMAP will use data from the selected contact property as input for the control and property information on the MSC.Marc CONTACT option. If a contact property is not chosen then the CONTACT option will not be written.

Note: You do not have to define contact pairs for MSC.Marc. If no contact pairs are present, and the Contact Property is selected in the Model Definition, FEMAP will export all contact segments to the CONTACT option, and all contact segments will be able to contact one another.

If you have defined contact pairs in FEMAP these will only be written to the CONTACT TABLE if the contact property is referenced under *Contact Property*. If no property is referenced then FEMAP will simply use the Contact Table defined in Section 8.6.1.5, "Contact Table"

Static Analysis

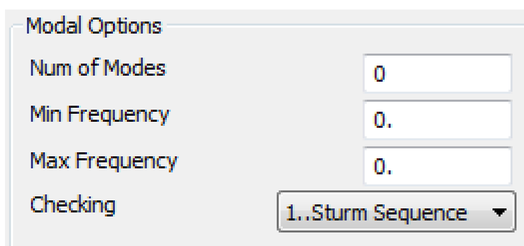
When performing a static analysis the Model Definition dialog box is the identical dialog box used for the History Definition.

If you do not need to define a history for your analysis then there is no need to define the History section. This is useful in linear analysis when all constraints and loading conditions have been applied in the Model Definition section.

The one major difference between the Model Definition and History Definition is that all options in the dialog box will not be available. The contact property information is only available in the Model Definition.

Modal Analysis Options

Modal Options are only be available for load cases defined in the History Definition (i.e., a “Case” in Analysis Set Manager).

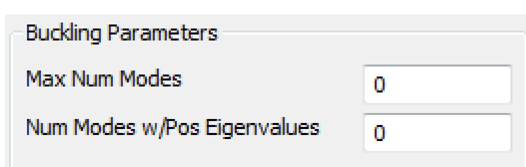


For modal analysis the eigenvalue extraction is obtained by creating History Definition and setting the Analysis Type to Modal. Then define the options for the MODAL SHAPE *Num of Modes, Min Frequency, Max Frequency, and set checking to either None or Strum Sequence.*

When Modal Analysis is defined for a History step then FEMAP will automatically create a second history which simply issues the Recover command to retrieve the modes

Buckling Options

Buckling Parameters will be available only in the Model definition and when Buckling has been set as the Analysis type.

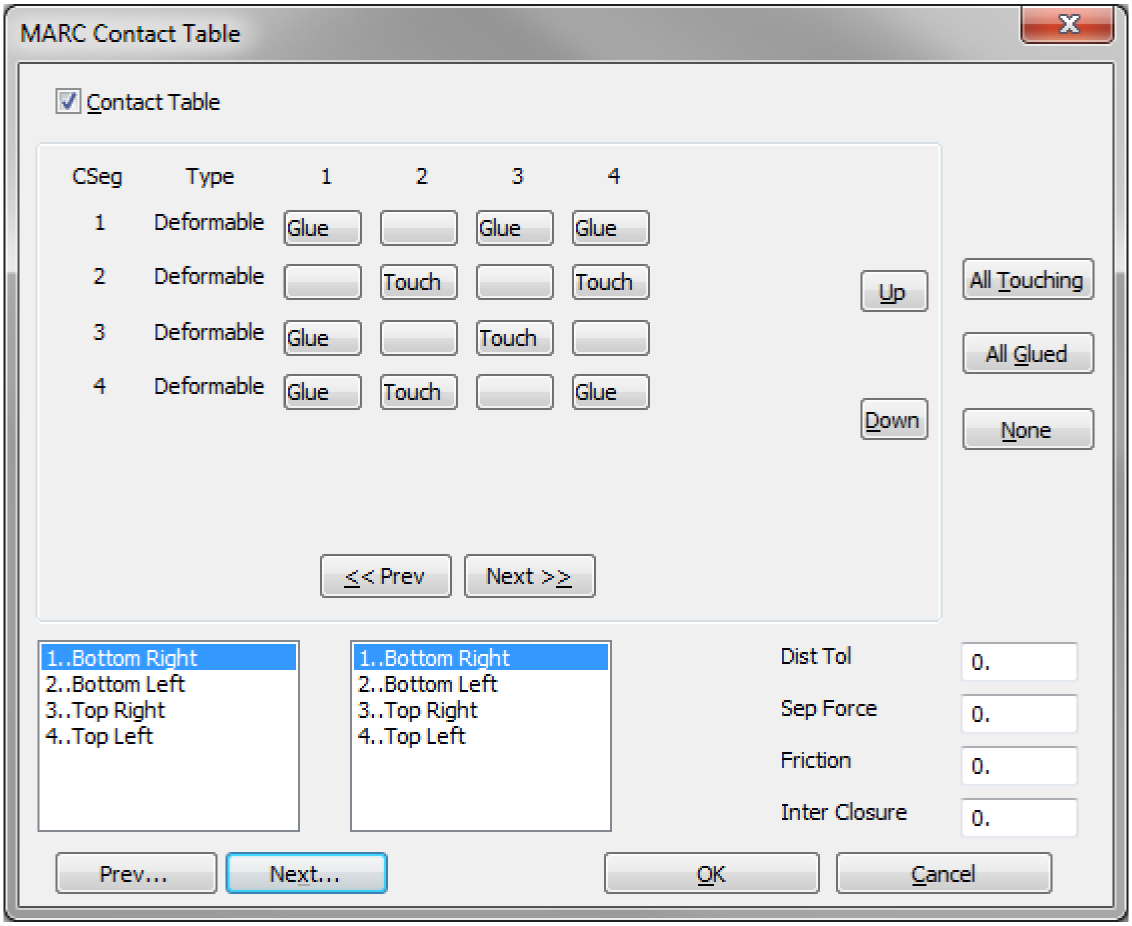


When the Buckling options are defined for the Model Definition FEMAP will write the BUCKLE parameter with the *Max Num Modes and Number of Modes w/ Positive Eigenvalues options*.

Then when you want to solve the buckling analysis simply create a History Definition with the Analysis Type set to Buckle. When buckling is defined in a History step then FEMAP will automatically create a second history which simply issues the Recover command to retrieve the buckling modes

8.6.1.5 Contact Table

The Contact Table allows a user to write a CONTACT TABLE command to activate or deactivate contact between specific contact segments. Begin by defining contact segments for each area that may be in contact and optionally define a contact property. The contact property can be chosen in the Marc Model Section 8.6.1.4, "Model Definition". This will write contact controls defined in the property to the CONTACT option.



The contact table can be defined for the model definition as well as individual history load steps. The table contains rows and columns that list existing contact segments in your model. Find the row that contains one of the contact segments that you want to put into contact, then find the column that contains the second contact segment that will be in contact. Simply press the button in the table where the two segments meet. Select the button to change the type of contact for this pair of contact segments. You have the choice of *Blank (No Contact) Glued or Tied*. You can make changes to the entire table by pressing the All Touching, All Glued or None buttons.

Parameters that are available for the pairs of contact segments are *Distance Tolerance, Separation Force, Friction, Interference Closure*. These can be defined for each pair of contact segments. The pair is automatically activated when you press the button in the table to change or activate contact for the pair of contact segments. The Parameters can also be set by selecting the two appropriate segments in the list boxes directly to the left of the parameters.

NOTE:

If the number of contact segments in the model is changed, the next time you enter the Contact Table dialog box FEMAP will fill in previously defined data and insert new empty cells to accommodate the new segments.

8.6.1.6 Boundary Conditions

The *Boundary Conditions* dialog box allows you to select the *constraint set, load set, and the initial conditions* load set for the Model Definition. Any constraints that are permanent in the model should be applied in a constraint set and chosen here. Often you may define only one constraint set for the analysis, but multiple loading conditions.

If all loading conditions are to be specified in the History Definition, the Loads section would be left 0..None for the Model Definition. The loads typically defined in the Model Definition would be those that resulted in only linear displacements, and then incremental loading would be applied in the History Definition to obtain the nonlinear results.

Initial conditions, such as initial displacement (INITIAL DISP), velocities (INITIAL VELO), and temperatures (INITIAL TEMP) should be created in their own load set and referenced in the *Initial Conditions* input box.

8.6.1.7 Output Requests

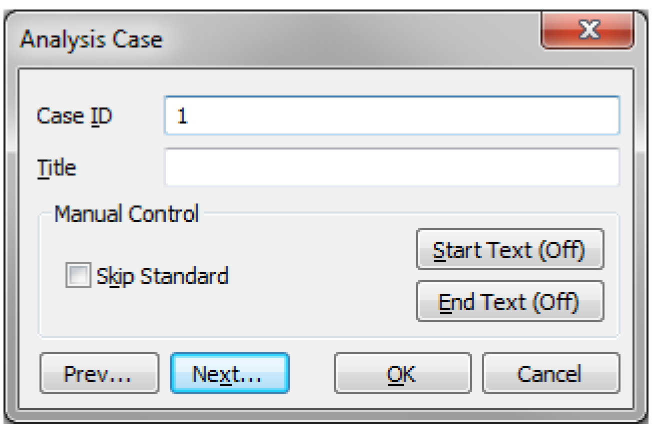
These options provide choices for the output files to request from MSC.Marc, as well as the type of output contained in the files. There are four choices: *Default, PostProcess Only, Print and PostProcess,* and *Print Only*. Either *Default, PostProcess Only* or *Print and Postprocess* must be selected to post-process results of the analysis in FEMAP. If *Postprocess Only* is chosen, a NO PRINT statement will be included in the Model Definition to prevent output of results to the printed (*.out) file.

You may also select what type of output to included in the files. If you select *Postprocess Only*, however, you will not be able to select any nodal outputs. MSC.Marc does not allow you to select the types of nodal output for the post-processing file. It automatically writes nodal information to the post file (*.t16 typically); therefore, these options are not applicable to the *Postprocess Only* option. Also, the individual elemental output requests are only applicable to the post-processing file. If a print option is chosen, the default elemental values will be written to the print file.

Note: Certain output requests such as heat flux and elemental temperature are not currently supported. These remain on the dialog box for future incorporation into the translator.

8.6.1.8 Creating Load Cases or History Definition

To create History Definition select *No Cases Defined* in the analysis set and either double click or press *New.*



The process is similar to the definition of the model definition. Define the Analysis Case dialog box. The *Case ID* is automatically determined.

***Manual* Control**

- If the *Skip Standard* switch is off, the software writes standard History step.
- Pick *Start Text* to add text to the beginning of the History step.
- Pick *End Text* to add text to the end of the History step.

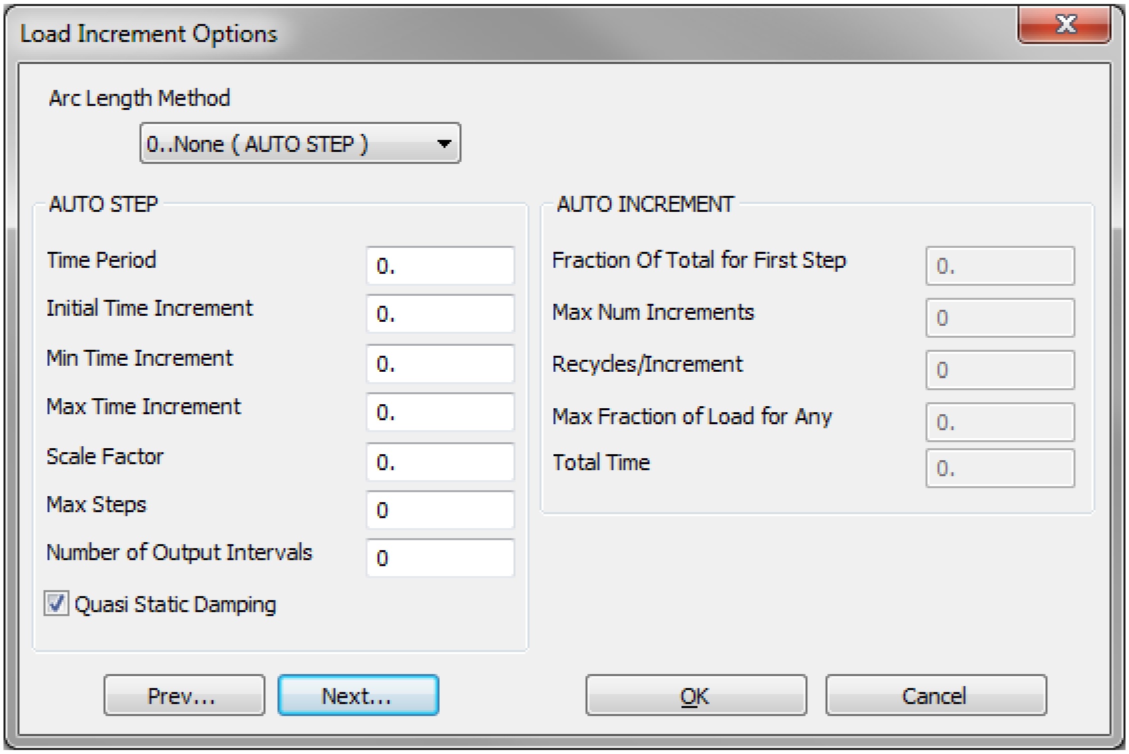
8.6.1.9 MARC History Definition

The History Definition contains the same options available in the Model Definition except that buckling is not available. See Section 8.6.1.4, "Model Definition"

Note: When selecting a new constraint set in the History Definition, the DISP CHANGE option will be written, removing all previous constraints in the model. Therefore, if you simply need to add constraints, the chosen constraint set must include all previous constraints, as well as new ones. This can be accomplished very easily in FEMAP by copying the Model Definition constraint set by using the *Model, Constraint, Copy* command, and then simply adding additional constraints to this set.

8.6.1.10 Load Increment Options

Load Increment options define the type of time stepping to be used for this load step.



Arclenght Method

If *arclength method* is set to 0..None (AUTO STEP) then the AUTO STEP option will be used for the analysis.

If the Arclength Method is set to *Crisfield, Riks, Modified Riks, or Crisfield/Modified Riks* then AUTO INCREMENT will be used.

As the Arclength Method is changed the respective control section will be ungrayed and for definition of the available options.

Note: The preferred time stepping method is to set the Arclength Method to 0..None (AUTO STEP).

8.6.1.11 Contact Table (History definition)

The contact table is defined just as it is in the model definition. See Section 8.6.1.5, "Contact Table"

8.6.1.12 Boundary Conditions (History Definition)

Boundary Conditions are defined just as they were in the Model Definition. See Section 8.6.1.6, "Boundary Conditions"

8.6.2 Writing an MSC.Marc Model with File, Export

Note: This method is obsolete and has been removed from the default configuration of FEMAP. If you need to use it for any reason, you must use the *File, Preferences* command, click the *Interfaces* tab, and check the option for “Enable Old Analysis Interfaces”. The use of this method for translation is NOT recommended. Use the *Model, Analysis* method instead.

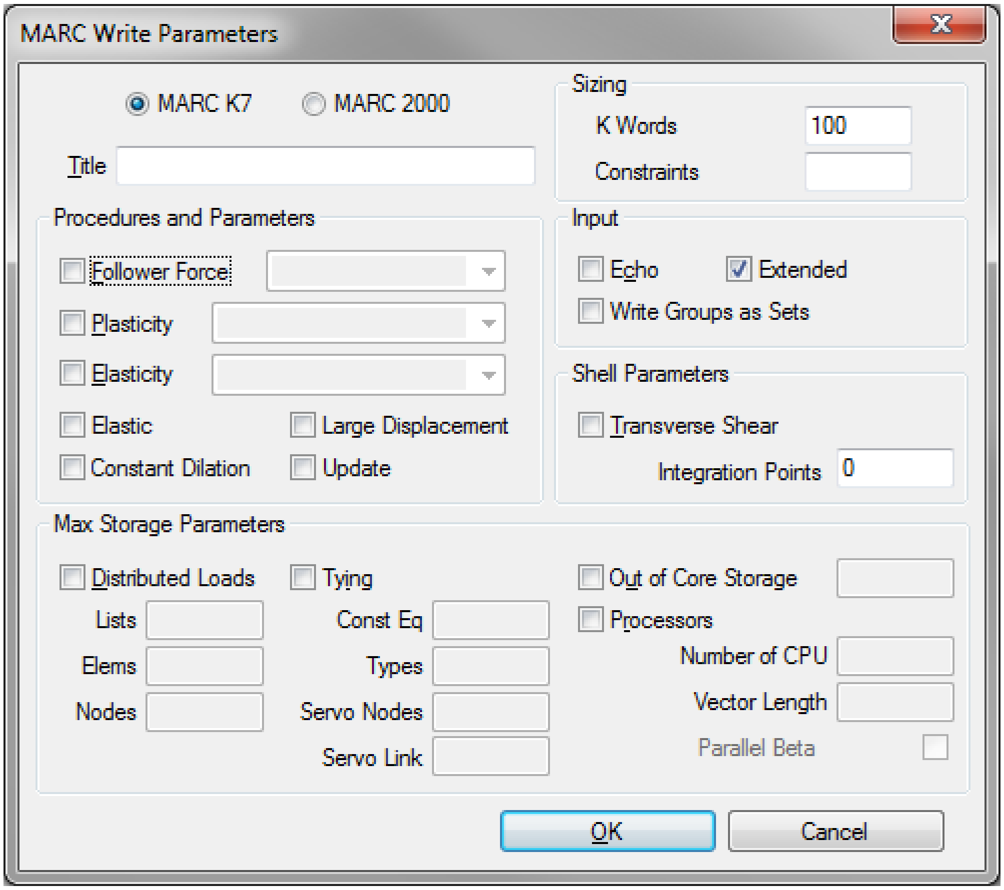
See Section 8.6.1, "Writing an MSC.Marc Model with Model, Analysis" for the most up to data method of writing a model for MSC.Marc.

The *File, Export, Analysis Model* command allows you to write your FEMAP model into a file that can be analyzed by MSC.Marc. Simply choose the *File, Export, Analysis Model* command, select *MARC*, and choose either *Static* or *Modes*t. Once you make these selections, you will see the standard file access dialog box, where you can specify the name of the file to be created. The default filename extension for this file is “.dat”, but you can choose any name.

The export process can be separated into three areas: analysis parameters, model definition, and history definition. Each of these areas are discussed more completely below.

8.6.2.1 *Analysis Parameters*

After choosing a file name, you will see the *MARC Write Parameters* dialog box. This dialog box controls the writing of all parameters to MSC.Marc. The specific options are described below.



MARC K7/MARC 2000

Use these buttons to choose your version of MARC.

Title

Enter a title for the analysis. It can be written to both binary and ASCII output files.

Sizing

This section determines the initial memory allocation (in thousands of Words - kWords) for the overall model, as well as constraints. The constraint value is not necessary unless you will have a History Definition in your model that contains more constraints than the Model Definition.

Input

This section allows you to obtain an *Echo* of the input in the output (*.out) file, write the *Extended* parameter for extended input, and to *Write Groups as Sets*. The *Extended* parameter will be necessary if any IDs in the model are above five digits. Without this parameter, all nodal, elemental, property, and material IDs must be under 100,000 or the analysis will fail due to input errors.

You may also choose to write all FEMAP groups as sets. This is a convenient method to obtain sets which you can then easily apply loading or other conditions by directly manipulating the MSC.Marc input deck.

Shell Parameters

This section determines whether the *Transverse Shear* option is chosen for a plate/shell analysis. You may also specify the number of integration layers through the thickness of plate elements. The default is 11 layers. Any input must be between 3 and 15, and be an odd number. If the model contains composites, this value is overridden by the number of composite layers.

Procedures and Parameters

This section enables you to select between many different parameters available in MSC.Marc.

Follower Force (Incremental Load)

This option controls whether follower forces are employed as well as whether loads applied on History Definitions are incremental or total. There are actually three different options involved with this one parameter: *Follower Force Stiffness* on or off, *Total* or *Incremental Loading*, and if *Follower Force* is on, (3) *Follower Force* is based upon *Displacement* at last iteration or beginning of increment. The one selection will set all three options.

Plasticity

This parameter controls the plasticity procedure that is used in MSC.Marc. There are currently five options for this parameter. They involve use of either the *Additive* or *Multiplicative Decomposition* method. If *Additive* is desired, you can also choose from two other options: *Mean Normal* or *Radial Return* method, and *Small or Large Strain* formulation. *Multiplicative Decomposition* requires radial return and large strain formulation. *Multiplicative Decomposition* is more accurate for large elastic and plastic strains, but it requires that all elasticity be isotropic. The default for this parameter if not set is *Additive Decomposition* using the mean normal method and small strain formulation.

Elasticity

This parameter controls the formulation for large strain elasticity. The default is a total Lagrange formulation. You may also select an updated Lagrange formulation. If you choose the total Lagrange formulation with Mooney or Ogden material models, the hybrid element formulation (Herrmann) must be used while the Foam material can use the standard elements. For updated Lagrange formulation, the standard elements amy be used for all three material types.

Elastic

This option can be implemented for elastic analysis with multiple loads. It essentially builds the stiffness matrix once, and then repeatedly back substitutes the load vectors to obtain the results. This option should not be used in nonlinear analysis, and will not be written if the *Large Displacement option* is checked.

Note: This option will also cause any constraint changes in the *History Definition* section to be ignored. Since the decomposition matrix is only formed once, at the end of the Model Definition stage (Increment 0), changes in constraints cannot be allowed.

Large Displacement, Update, Constant Dilation

These options invoke the *Large Displacement* option, the updated Lagrange procedure (instead of total), and *Constant Dilation*. Constant dilation is recommended for lower order elements in nearly incompressible analysis. This option can also be used for the individual elements by specifying a hybrid element formulation for the lower order standard element types (bricks, plane strain, and axisymmetric).

Max Storage Parameters

This section controls memory allocation for several different lists. Many of these parameters will only be required if a History Definition has a larger number of lists than the model definition.

Distributed Loads

This parameter controls the memory allocation for elemental and nodal loading. Again, this is only required if you will be applying more loads in a History Definition than was originally on the Model Definition inputs include the total number of lists for distributed loads, the maximum number of elements in a DIST LOADS list (pressure, distributed load, gravity), and the maximum number of nodes in a POINT LOAD list. FEMAP will export each load as its own list, therefore, the number of lists must be equal to or greater than the total number of loads. The number of nodes and elements in any specific list, however, will typically be 1 since each load has its own list containing one member. The one exception to this is FEMAP body loads (gravity and rotational velocity), which will contain the total number of elements in the model in one list.

TYING

This parameter allocates storage for tying data which includes FEMAP rigid elements (MARC Tying Constraints 1-6, 100) and constraint equations (MARC Servo Links).

The maximum number of MARC constraint equations should be set equal to the number of FEMAP rigid elements in the model, and the number of different types can be safely set to 7 (only 1-6 and 100 are currently supported).

The *Servo Nodes* should be set to the maximum number of nodes in any constraint equation (set to 2 if no FEMAP constraint equations), and the *Servo Link* should be set to the maximum number of FEMAP constraint equations.

Out of Core Storage

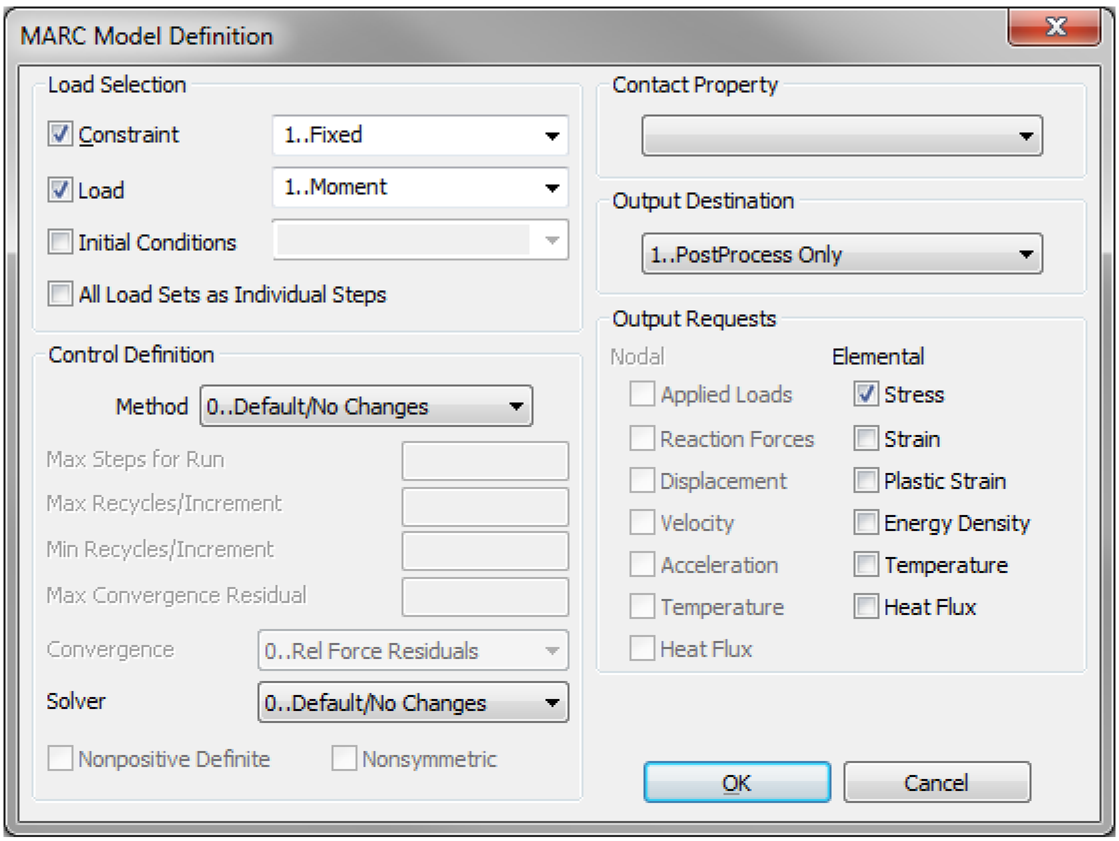
This option invokes the ELSTO parameter for large models. It will cause all elemental quantities to be stored in an auxiliary area (unit 3) to free core memory for calculations. The input for this option is in Words.

Processors

This option enables you to specify the number of CPUs, the vector length and whether the beta matrices are to be formed in parallel.

8.6.2.2 Model Definition

Once you press *OK* on the *MARC Write Parameters* dialog box, the *MARC Model Definition* dialog box will appear. This dialog box is separated into four major sections: *Load Selection*, *Control Definition, Contact Property,* and *Output Requests*.



Load Selection

The *Load Selection* area allows you to select the constraint set, load set, and the initial conditions load set for the Model Definition. Any constraints that are permanent in the model should be applied in a constraint set and chosen here. Often you may define only one constraint set for the analysis, but multiple loading conditions.

If all loading conditions are to be specified in the History Definition, the load set would be left unchecked for the Model Definition (in which case you would have had to input values for the *Distributed Loads* parameter on the previous dialog box to prevent a fatal error). The loads typically defined in the Model Definition would be those that resulted in only linear displacements, and then incremental loading would be applied in the History Definition to obtain the nonlinear results.

Initial conditions, such as initial displacement (INITIAL DISP), velocities (INITIAL VELO), and temperatures (INITIAL TEMP) should be created in their own load set and referenced in the *Initial Conditions* input box.

The final option, *All Load Sets as Individual Steps*, will simply write every load step as an individual History Definition and will provide no opportunity to change any inputs for the History Definition. If a load set is chosen in the Model Definition, it will not be repeated in a History Definition.

Control Definition

This section determines the type of solution method (including the convergence criteria), and the solver type.

Method

For the solution method, you can simply select the default, or choose from either the Newton Raphson techniques (Full, Modified, or Strain Correction) or the Secant method. If you choose a method, you can then also specify bounding parameters for the number of recycles, as well as the maximum steps for a run. If no values are entered, the defaults will be used.

You may also define the convergence criteria as well as choose the *Nonpositive Definite* option. You have six options in this box, which are really the combination of two outputs defining the basis of convergence: F*orce, Displacement,* or *Strain Energy* for either relative or absolute values. The selection of the *Nonpositive Definite* option will force solutions of problems with a nonpositive definite matrix.

Solver

This option allows you to chose from several different direct and iterative solvers. When this option is active, you can also select options for *Nonpositive Definite* and *Nonsysmmetric* matrices.

Contact Property

All contact segments in MARC are exported to the CONTACT option in MSC.Marc. This option includes contact property information for all contact entities contained in the table. To define these values, simply create a contact property in FEMAP, and then select this property under *Contact Property*. If there are contact segments in the model, FEMAP will use data from the selected contact property as input for the control and property information on the MSC.Marc CONTACT option.

Note: You do not have to define contact pairs for MSC.Marc. If no contact pairs are present, FEMAP will export all contact segments to the CONTACT option, and all contact segments will be able to contact one another. If you want to limit contact between the contact segments, simply define contact pairs. FEMAP will export the contact pairs to a CONTACT TABLE. Once the CONTACT TABLE option is invoked in MSC.Marc, all contacts are removed, so only those segments placed in contact pairs will be checked for contact.

Output Destination and Requests

These options provide choices for the output files to request from MSC.Marc, as well as the type of output contained in the files. There are three choices: *PostProcess Only, Print and PostProcess,* and *Print Only*. Either *PostProcess Only* or *Print and Postprocess* must be selected to post-process results of the analysis in FEMAP. If *Postprocess Only* is chosen, a NO PRINT statement will be included in the Model Definition to prevent output of results to the printed (*.out) file.

You may also select what type of output to included in the files. If you select *Postprocess Only*, however, you will not be able to select any nodal outputs. MSC.Marc does not allow you to select the types of nodal output for the post-processing file. It automatically writes nodal information to the post file (*.t16 typically); therefore, these options are not applicable to the *Postprocess Only* option. Also, the individual elemental output requests are only applicable to the post-processing file. If a print option is chosen, the default elemental values will be written to the print file.

Note: Certain output requests such as heat flux and elemental temperature are not currently supported. These remain on the dialog box for future incorporation into the translator.

8.6.2.3 History Definition

Once the Model Definition has been written, a similar dialog box will appear for the History Definition. You can select different constraints and load sets for the History Definition, as well as change solution and solver options. Slightly different dialog boxes will appear based upon whether you are exporting for static or modal analysis.

Hint: When performing a contact or highly nonlinear analysis, it is best to define the initial step size and maximum number of increments. This can be done in the load step used for the History Definition under *Model, Load, Nonlinear Analysis, Static* type. Set the *Number of Increments* (FEMAP calculates the initial time step as the reciprocal of this number) and the *Max Iterations/Step*. If you have not defined these values, FEMAP will default to an initial step size of 0.1 and maximum increments of 100 for any analysis involving contact.

Static Analysis

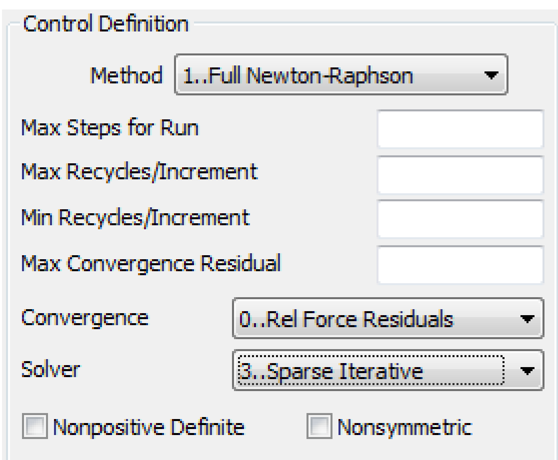
When performing a static analysis, the identical dialog box used for the Model Definition is employed for the History Definition. If you do not need to define a history for your analysis, simply press *Cancel* and FEMAP will close the file. This is useful in linear analysis when all constraints and loading conditions have been applied in the Model Definition section.

The one major difference between the Model Definition and History Definition is that all options in the dialog box will not be available. Initial conditions and contact property information are unique to the Model Definition, therefore these options will be unavailable. Furthermore, the solution method, solver, and output options will default to *No Change*. Thus, if you want to keep the current options but want to change the constraints or loading conditions, simply select the constraint and load sets and press *OK*. The loading conditions can be either incremental or total, based upon the Follower Force parameter input in the *Parameters* section (Default is Incremental).

You can define multiple histories by selecting the desired options and then pressing *OK*. The same dialog box will reappear but the *History Definition Number* on the dialog box is incremented. When you have defined all the desired histories, press *Cancel* to end the translation.

Note: When selecting a new constraint set in the History Definition, the DISP CHANGE option will be written, removing all previous constraints in the model. Therefore, if you simply need to add constraints, the chosen constraint set must include all previous constraints, as well as new ones. This can be accomplished very easily in FEMAP by copying the Model Definition constraint set by using the *Model, Constraint, Copy* command, and then simply adding additional constraints to this set. Remember, however, if the total number of constraints is larger in a History Definition than in the Model Definition, you will need to specify the total number of constraints on the SIZING option contained earlier on the *MARC Write Parameters* dialog box.

Modal Analysis



Modal analysis has a slightly different input than static. The *Control Definition* section is modified to include the *Min* and *Max Frequency*, *Number of Modes*, and whether to perform Sturm Sequence *Checking*. All other inputs are similar to static analysis.

When you press *OK* for the first history, FEMAP will export this history, as well as create a second history which simply issues the Recover command to retrieve the modes. You can export multiple histories for modal analysis by continuing to specify conditions and pressing *OK*. *Cancel* will end the input.

8.6.3 Performing an MSC.Marc Analysis

FEMAP can launch MSC.Marc and automatically run your analysis, either from the File Analyze command or pressing Analyze in Model Analysis, if you first setup the analysis program. To do this, before running FEMAP, you must establish an environment variable named MARC_EXE. Set the value of this environment variable to the full path to the location of the files used to start the MSC.Marc solver executable, for example:

```
SET MARC_EXE = c:\marc\marc2001\tools
```

Note that this does not contain the name of the file used to start MSC.Marc. FEMAP automatically looks for the file “run_marc.bat” in the directory that you specify. Once this environment variable has been defined, FEMAP will be able to launch the analysis program and monitor the job until it is complete.

NOTE: Once the analysis has been launched the Analysis Monitor will appear and monitor the job status. Section 4.10.2.2, "Analysis Monitors"

8.6.4 Reading an MSC.Marc Analysis Model

In addition to the ability to write MARC models and read results, FEMAP can also read the nodal locations and element connectivity from the MSC.Marc binary post-processing files (typically *.t16). There are two ways to read this information. First, you can choose *File Import, Analysis, Model*, and then choose the output file that you want to read. In addition, if you start with an empty model, and directly select *File, Import, Analysis Results*, FEMAP can read both the model and results in one pass.

8.6.5 Post-processing MSC.Marc Results

When you have completed your MSC.Marc analysis, you can load results into FEMAP for post-processing. You should typically start with a FEMAP model that corresponds to the MSC.Marc analysis; however, FEMAP will read the nodes and elements form the post-processing file if your model does not contain any nodes or elements.

To load the results, choose the *File, Import, Analysis Results* command and select *MARC*. FEMAP will display the standard file access dialog box so you can choose the file you want to post-process. FEMAP reads the binary post-processing file, typically *.t16. Once you select the file, FEMAP will automatically read and store the results. No further input is required. Also, if the post-processing file contains information which FEMAP cannot process, such as die information or NURB rigid surfaces, FEMAP will issue an error message and quit reading. In these cases, very little data may be recovered.

8.7 Nastran Interfaces

FEMAP provides direct interfaces to the NX Nastran, MSC/MD Nastran and Autodesk Nastran (formally NEi/Nastran) model and output file formats. Topics include:

- Section 8.7.1, "Writing a Nastran Model with Model, Analysis"
- Section 8.7.2, "Writing a Nastran Model with File, Export" (Obsolete)
- Section 8.7.3, "Performing a Nastran Analysis"
- Section 8.7.4, "Reading Nastran Models"
- Section 8.7.5, "Post-processing Nastran Output"

For more information on the entities that are translated, see Section 7.1, "Translation Table for ANSYS, I-DEAS, NASTRAN, and MSC Patran".

Nastran from Other Vendors

If you are using COSMIC NASTRAN, CSA/NASTRAN, ME/NASTRAN, SSS/NASTRAN, or UAI/NASTRAN, you must use the *File, Export, Analysis Model* command, then select the FEMAP interface that is specifically for that vendor. For more information, see Section 8.7.2, "Writing a Nastran Model with File, Export".

You may have to make slight changes to the files that FEMAP produces or requires to make them compatible with the version that you are using. Depending on how closely your vendor's Nastran output matches MSC.Nastran output, you may have to make more significant changes before you can post-process.

For more information, see the analysis program vendor's documentation.

8.7.1 Writing a Nastran Model with Model, Analysis

You can write the FEMAP model into a file that can be read by Nastran through either the *Model, Analysis* or the *File, Export, Analysis Model* commands. Although the commands have a different user interface, they both produce a file that contains the three required sections: Executive Control, Case Control, and Bulk Data.

Once you select the command, you then select the appropriate version of Nastran:

NX Nastran, MSC Nastran, or Autodesk Nastran (NEi/Nastran).

- The NX Nastran interface supports NX Nastran version 10.2 and earlier.
- The MSC Nastran interface supports MSC.Nastran versions 2014 and earlier.
- The NEi/Nastran interface supports version Autodesk Nastran 2016 and earlier.

The remainder of this section describes the steps for defining a Nastran analysis with *Model, Analysis*. You will:

1. Define an analysis set. (See Section 8.7.1.1, "Analysis Set".)
2. Define executive and solution options. (See Section 8.7.1.2, "NASTRAN Executive and Solution Options").
3. Define bulk data options. (See Section 8.7.1.3, "NASTRAN Bulk Data Options".)
4. Define output requests and boundary conditions. (See Section 8.7.1.6, "Master Requests and Conditions", Section 8.7.1.7, "Boundary Conditions" and Section 8.7.1.8, "NASTRAN Output Requests").
5. Optionally, define options for analysis types (See Section 8.7.1.9, "NASTRAN Modal Analysis", Section 8.7.1.10, "NASTRAN DDAM Analysis (Modal Analysis Only)", Section 8.7.1.11, "NX Nastran Rotor Dynamics (SOL 110 and 111 Only)", Section 8.7.1.12, "NASTRAN XY Output for Modal Analysis", Section 8.7.1.13, "NASTRAN Direct Transient Analysis", Section 8.7.1.14, "NASTRAN Modal Transient Analysis", Section 8.7.1.15, "NASTRAN Direct Frequency Analysis", Section 8.7.1.16, "NASTRAN Modal Frequency Analysis", Section 8.7.1.17, "NASTRAN Response Spectrum Analysis", Section 8.7.1.18, "NASTRAN Random Response Analysis", Section 8.7.1.19, "NASTRAN Design Optimization Options", Section 8.7.1.20, "Heat Transfer Nonlinear Control Options", Section 8.7.1.21, "Nonlinear Static, Nonlinear Transient, and Creep", Section 8.7.1.22, "Special Notes for Models with Axisymmetric Elements", Section 8.7.1.23, "Special Notes for Heat Transfer Analysis", Section 8.7.1.24, "Advanced Nonlinear Analysis (NX Nastran Only)", and Section 8.7.1.25, "Advanced Nonlinear Explicit (NX Nastran Only)", Section 8.7.1.26, "Static Aeroelasticity Analysis", Section 8.7.1.27, "Aerodynamic Flutter Analysis", Section 8.7.1.28, "Contact Parameters (MSC Nastran Only)", and Section 8.7.1.29, "Superelement Analysis"),
6. Optionally, define additional cases that include different load sets and constraint sets. (See Section 4.10.1.6, "Cases" in *FEMAP Commands Manual*.)

8.7.1.1 Analysis Set

When you create an analysis set, you define the solution parameters, boundary conditions, and output for the Nastran analysis. An analysis set is stored with the model file, so you can reuse this data. You can also store an analysis set in a FEMAP analysis library.

To define an analysis set:

1. Pick *Model, Analyze* to open the *Analysis Set Manager* dialog box.
2. Pick the first item on the list, then pick *New*. (You can also double-click on the item).
3. Choose the *Analysis Program* and *Analysis Type*. These options determine which element types (structural or heat transfer) will be written, and set the defaults for the other NASTRAN interface dialog boxes.

   You can also choose to use the *Integrated Solver* (NX Nastran only), a *Linked Solver*, or *VisQ*. See Section 2.6.2.6, "Solvers" or Section 4.10.2.1, "Run Analysis Using Linked Solver / VisQ / Local Settings" in the FEMAP Commands Manual for more information.
4. The best way to define an analysis set is to pick *Next* to work through the dialog boxes in order. Alternatively, you can pick *OK* to close the dialog box. From the analysis set list, you can then doubleclick on a parameter to bring up the dialog box.

For more information about using the Analysis Set Manager, see Section 4.10, "Preparing for Analysis" and more information about defining an Analysis Set, see Section 4.10.1, "Defining a Analysis Set".

8.7.1.2 NASTRAN Executive and Solution Options

The *Executive and Solution Options* dialog defines the *Executive Control, Solution Control, Restart Control,* and *Manual Control* text options for your Nastran model.

Solver

Direct Output To

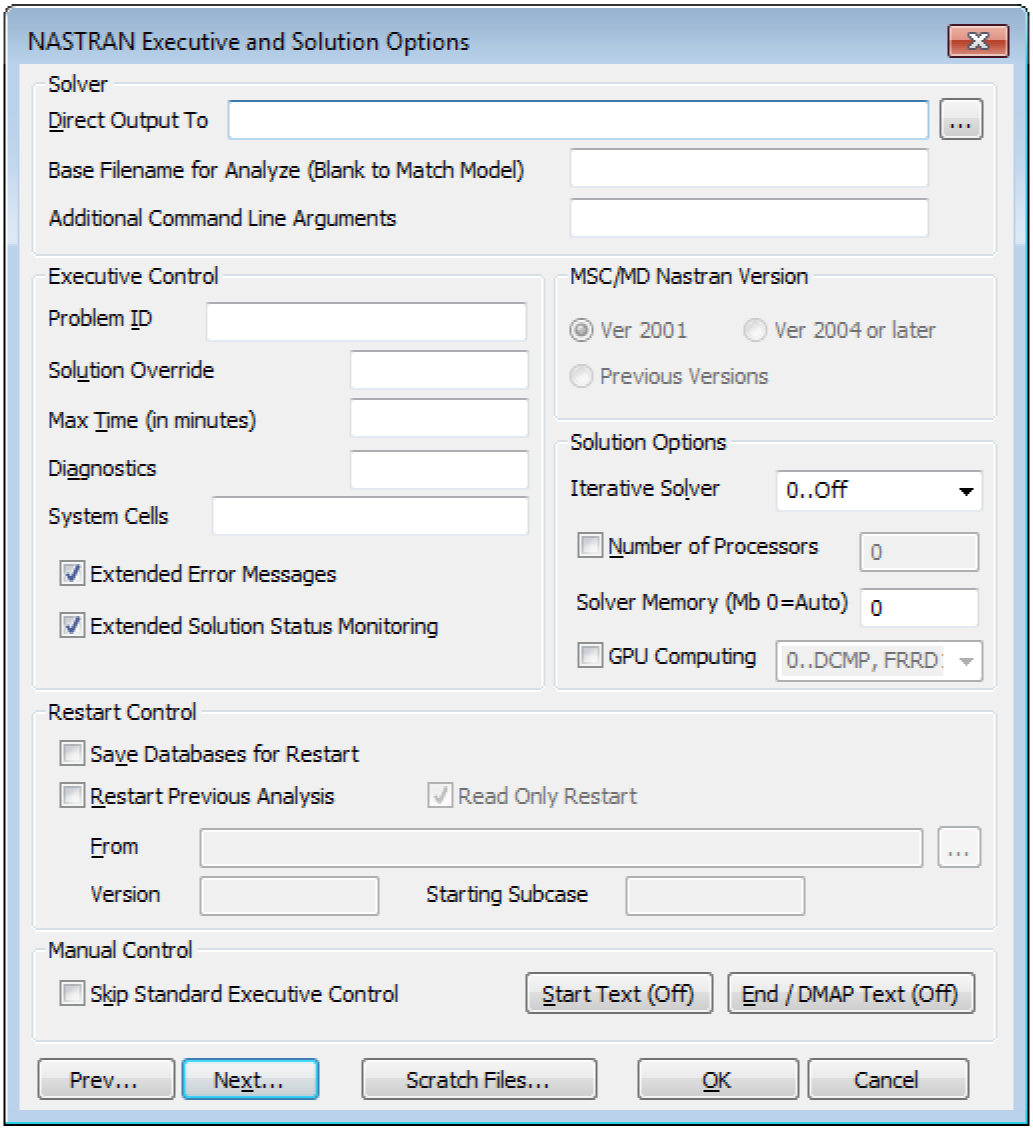
This option is used to specify a location for the Nastran output. Click "..." button to browse to a directory.

Base Filename for Analyze (Blank to Match Model)

If the field is left blank the model name will be used as the 'base name' for the Nastran input file. If any text exists in this field, that text will be used as the 'base name'. The 'base name' and a 3 digit suffix which increments each time the input file is written to the same directory are used as the input file name (i.e., 'base name'###.dat).

Additional Command Line Arguments

Allows you to enter any additional command line arguments which are not supported through the FEMAP user interface. If anything is specified in the *Arguments* field on the *Solvers* tab of *File, Preferences*, for your version of Nastran, it will also appear in this field. Any command line arguments entered override any other settings, even those found in the Nastran configuration file (*.rcf file). For more information about using the *Solvers* tab, see Section 2.6.2.6, "Solvers" in the *FEMAP Commands Manual* and/or consult the documentation for your version of Nastran to determine available command line arguments to include when launching the solver.



Executive Control

Problem ID

Written as a title to the ID command.

Solution Override

Selects the DMAP “solution sequence” that will be executed. FEMAP will automatically define this as SESTATIC, SEMODES, SEDTRAN, SEMTRAN, SEDFREQ, SEMFREQ, SEBUCKL, NLSTATIC, NLTRAN, NLSCSH, NLTCSH, AESTAT, SEFLUTTER, “601,106”, “601, 129”, or “701”, but you can change it to any of the solution sequences that you want to use.

Max Time (In Minutes)

Sets the maximum allowable CPU time for this analysis. Do not set this number too low, or your analysis will terminate prematurely. The TIME statement is optional, so if no value is specified, the TIME statement is not written.

Diagnostics

Allows you to specify any diagnostic lines.

System Cells

Allows you to specify Nastran System Cells using the NASTRAN statement. The must be entered as: SYSTEM ("system cell #")="# of option for specified system cell". For example, Extended Error Messages is SYSTEM (319)=1

Extended Error Messages

When enabled, option prints out the extended Nastran error messages to the .f06 file. This will assure that FEMAP is always using the most current error messages from your version of Nastran. These extended error messages will also be available when using the Analysis Monitor with the interlocked version of FEMAP with NX Nastran. (writes SYSTEM (319) = 1 to the Nastran file)

Note: It is highly recommended to use the *Extended Error Messages* option when using the Nastran solver. There is then only one screen used to review Nastran Error Messages. If you do not use this option, clicking the *Help* button in *Message Review Details* dialog box will link to an older file of Nastran Error Messages which may or may not be accurate for newer Nastran versions.

Extended Solution Status Monitoring

When enabled, option allows the *NX Nastran Analysis Monitor* in FEMAP to receive additional feedback from the solver. The type of feedback will be determined by the type of analysis currently being monitored (For instance, a static analysis will return "Sparse Matrix Solver" information about the number of equations which to be solved in total and then give updates on how many have been solved so far),

Note: This feedback is only retrieved from NX Nastran every time the *NX Nastran Analysis Monitor* is updated, which is once every 5 seconds. Also, the feedback will only start after the number of equations has been determined, which will differ for each analysis job

MSC/MD NASTRAN Version

Select the version that you are using:

- ***Ver 2004 and Above*** (MSC.Nastran 2004 and above), ***Ver 2001*** (MSC.Nastran 2001), or ***Previous Versions*** (up to version 70.7).

Solution Options

Iterative Solver

Specify whether to use the iterative solver or element iterative solver (NX Nastran only).

Number of Processors

Allows you to set the number of processors.

Solver Memory (Mb 0=Default)

Allows you to allocate the amount of memory for Nastran to use when solving. If you leave this field blank, Nastran will use the value currently set in your Nastran resource file (Nast*.rcf located in the "conf" directory), which by default is often set to "memory = estimate" (NX Nastran estimates how much memory the job requires). This is usually recommended. The mechanism FEMAP uses to set this option is to add a command line option (memory = VALUE mb) when the job is submitted. This will override the value currently set in your Nastran Resource file.

Note: Please refer to NX Nastran documentation for more information on setting the correct memory value for the solver. Allocating more memory than your machine has can cause the solver to fail and setting this value too low can cause the solver to be less efficient.

GPU Computing

When enabled, has NX Nastran automatically determine if a device with GPUs exists and, if so, how to use it during the solve. Three options exist which will write different arguments to the Nastran command line.

- **"0..DCMP, FRRD1"** - includes "gpgpu=any" on command line, enables all GPU module computations.
- **"1..DCMP"** - includes "cl_dcmp=1" on command line, enables only DCMP GPU module computations.
- **"1..FRRD1"** - includes "cl_frrd=1" on command line, enables only FRRD1 GPU module computations.

Restart Control

Save Databases for Restart

When enabled, the *.MASTER and *.DBALL files will be saved to allow for a restart analysis.

Restart Previous Analysis and options

When *Restart Previous Analysis* is enabled, the *Read Only Restart* option and the *From*, *Version*, and *Starting Subcase* fields become available.

From

Use this field to specify a directory path to the *.MASTER file (use the ".." button to browse to find the file).

Read Only Restart

Enabling this option will force Nastran to attach to the *.DBALL file in "Read Only" mode, which will prevent the *.DBALL from possibly becoming corrupted during the restart.

Version and Starting Subcase

For more experienced users, use the *Version* field to specify the version of the *.DBALL file (blank is default, which is "last", otherwise must be an integer value) and/or use the *Starting Subcase* field to specify which subcase to restart from for a nonlinear analysis.

Manual Control

Skip Standard Executive Control

When off, the software writes standard Executive Control. If this switch is on, the software writes ONLY the contents of *Start Text* or *End Text* that you enter.

Start Text

Click to add text to the beginning of the Executive Control section. This capability can be used to include standard DMAP alter sequences, Job Control (JCL) statements, or other standard modifications to the beginning of your Nastran file. In addition, the *Create ASSIGN* button allows for creation of various types of ASSIGN statements via the *Create ASSIGN Statement* dialog box.

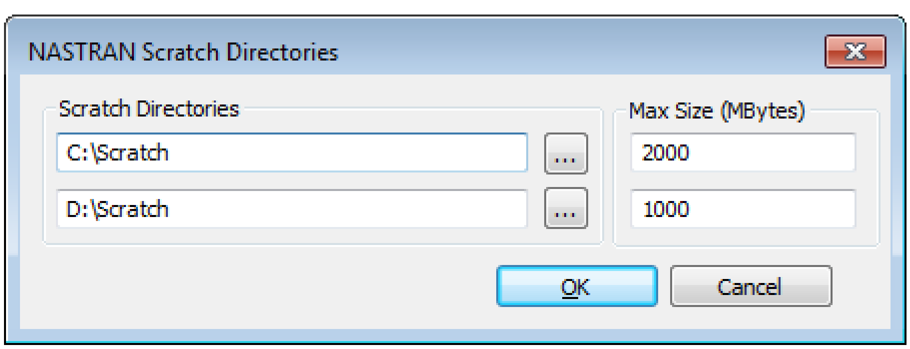
End/DMAP Text

Click to add text to the end of the Executive Control section. This is often where any DMAPs should be included with your Nastran input file.

Scratch Files... button

Sometimes it may be necessary to "break-up" a large Nastran "Scratch File" into different pieces and place each piece into a specific directory. This insures the job has enough disk space in which to write the "scratch file". Usually, you would use this to place different parts of the Scratch file onto separate hard drives (i.e. the "C:" and "D:" drive). This button will take you to the *NASTRAN Scratch Directories* dialog box, which allows you to specify *Scratch Directories* by entering the path directly or browsing to it (use the *"..."* button).

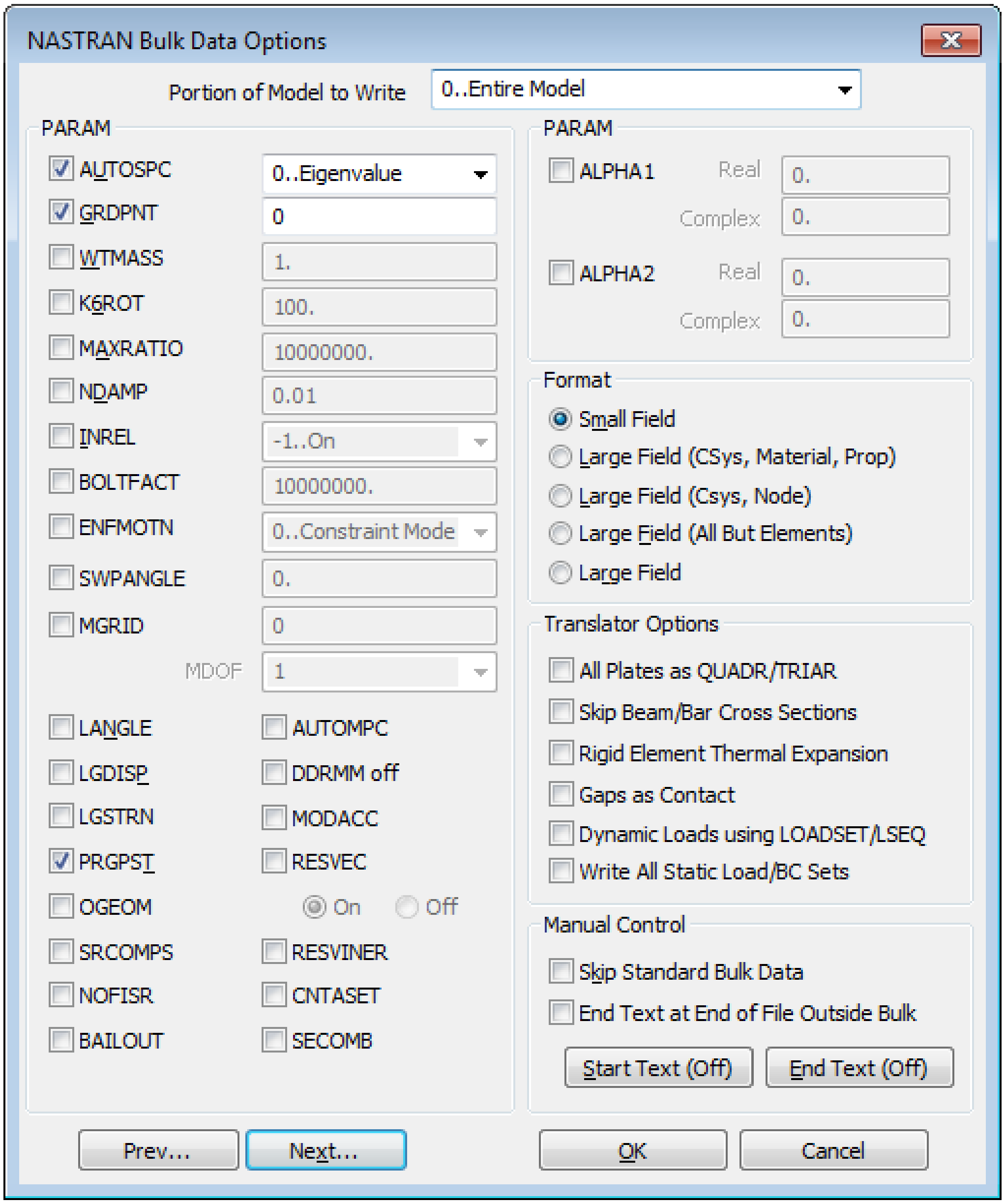
You must also enter a *Max Size* (in Megabytes) for each scratch file. Once the *Max Size* has been reached for the first specified scratch file, Nastran will being writing to the next specified scratch file. This will write the appropriate INIT SCRATCH LOGICAL, ASSIGN SCR, and ASSIGN SCR1 entries to the Nastran deck.



Note: Please be sure the total value of the two *Max Size* fields is larger than the Nastran "Scratch File" or Nastran could experience issues.

8.7.1.3 NASTRAN Bulk Data Options

Use the *Bulk Data Options* dialog to define only export a potion of the model to write out to Nastran and/or specify PARAM cards, format, and translator options.



Portion of Model to Write

Allows you to select a previously defined group, then only exports the supported entities in that group to the Nastran input file. In some cases, exporting elements without associated nodes or material/property entries may be desired, but this will create an input file which cannot be run by Nastran unless additional entities are added to the file or the file is later referenced by an INCLUDE statement in another Nastran input file.

Note: If a “ready to run” input file is desired, it may be helpful to use the *Group, Operations, Add Related Entities* command on the group before selected with this option. See "Group, Operations, Add Related Entities..." in Section 6.4.3.2, "Group, Operations Menu" of the F*EMAP Commands Manual* for more information.

PARAM

FEMAP will write PARAM cards for those options selected. If you want to use additional parameters besides those listed, you can add them with the *Start Text* button.

AUTOSPC - you can also control the format of the AUTOSPC command to the Nastran convention (PARAM, AUTOSPC, YES). In NX Nastran you have a choice to use the default method (0..Eigenvalue) or a method available in NX Nastran version 4.1 and higher (1..Singular Value Decomposition). If you use the “SVD” method, FEMAP will write a system cell to NX Nastran (SVDSPC=1).

GRDPNT - when enabled with value >1, writes PARAM,GRDPNT,(Node ID), which causes the grid point weight generator to be executed using the specified node ID as the “reference point”. When enabled with value = 0, which is the default in FEMAP, writes PARAM,GRDPNT,0, which causes the grid point weight generator to be executed using the origin of the basic coordinate system as the “reference point”. When disabled, writes PARAM,GRDPNT,-1, which suppresses the computation and output of data from the gird point weight generator. Additional information about using this PARAM in certain type of analysis is available in the Nastran Quick Reference Guide.

WTMASS - when enabled, writes PARAM,WTMASS,(value). The terms of the structural mass matrix are multiplied by the entered value. This is sometimes used to convert mass to weight and vice versa.

K6ROT - when enabled, writes PARAM,K6ROT,(value). Specifies the stiffness to be added to the normal rotation for CQUAD4 and CTRIA3 elements. Default value is 100.0 and parameter is ignored by CQUADR, CTRIAR, CQUAD8, and CTRIA6 elements.

MAXRATIO - when enabled, writes PARAM,MAXRATIO,(value). The ratios of terms on the diagonal of the stiffness matrix to the corresponding terms on the diagonal of the triangular factor are computed. If, for any row, this ratio is greater than MAXRATIO, the matrix will be considered to be nearly singular (i.e.,has mechanisms). Default value is 1.0E7.

NDAMP - when enabled, writes PARAM,NDAMP,(value). Specifies value for numerical damping used by the TSTEPNL entry to achieve numerical stability when METHOD=”ADAPT” in Nonlinear Transient (SOL 129) or Transient Heat Transfer (SOL 159). A value of 0.0 means there will be no numerical damping. Default value is 0.01 and recommended range of values is from 0.0 to 0.1.

INREL - controls the calculation or inertia relief or enforced acceleration in Statics (SOL 101), Buckling (SOL 105), and Optimization (SOL 200). Using “-1..On” requests that inertia relief or enforced motion be performed and requires SUPORT entries. Using “-2..Automatic (Statics)” can only be used for statics and requests automatic inertia relief, in which case SUPORT entries are not necessary.

BOLTFACT - used to reduce the bolt stiffness during the first phase of a bolt preload analysis (NX Nastran only).

ENFMOTN - for NX Nastran only and is used to control which formulation is used for enforced motion analysis (and mode acceleration, if requested). This writes out ENFMOTN (i.e., SYSTEM(422)) to the NASTRAN line. Choosing “0..Constraint Mode” uses the constraint method of enforced motion formulation (and new mode acceleration method), “1..Absolute” uses the absolute displacement enforced motion formulation (and old mode acceleration method), and “2..Absolute, Viscous Damping” uses the absolute displacement enforced motion formulation, but includes a modal viscous damping coupling term to improve accuracy.

SWPANGLE - the angular increment in degrees at which failure indices and strength ratios are computed and output for laminates in direct frequency (SOL 108) and modal frequency (SOL 111) analysis (NX Nastran Only).

MGRID and ***MDOF***- used to specify a specific node (MGRID) and degree of freedom (MDOF = 1, 2, 3, 4, 5, or 6) to monitor during a direct frequency or direct transient response, plotted in the *NX Nastran Analysis Monitor*.

LANGLE - when enabled, writes PARAM,LANGLE,2, large rotations in nonlinear analysis are determined using a Rotation Vector method, when geometric nonlinearity is defined (PARAM,LGDISP,1). When disabled, large rotations are computed with the gimbal angle method.

LGDISP - when enabled, which is the default when creating a new analysis set for nonlinear analysis, writes PARAM,LGDISP,1. When running a nonlinear analysis, all elements which have a large displacement capability are assumed to have large displacement effects (i.e., updated nodal coordinates and follower forces). When disabled, then no large displacement effects will be considered.

LGSTRN - when enabled, writes PARAM,LGSTRN,1. Only used by the NX Nastran advanced nonlinear analysis types (SOLs 601 and 701) and assumes large strains, displacements, and rotations (also, automatically sets PARAM,LGDSIP,1). Large strain formulation only applicable to 2D Axisymmetric, plane strain, 3D solid, and single layer shell elements.

PRGPST - when enabled, which is the default, singularities removed by AUTOSPC are written to the f06 file. When disabled, printout of singularities is suppressed, expect when they are not going to be removed.

OGEOM - when enabled, writes specific geometry blocks to the results file, depending on the option selected for PARAM,POST. See entry for PARAM,POST in Nastran Quick Reference Guide for more information.

SRCOMPS - controls the computation and printout of ply strength ratios. When 'on', ply strength ratios are output for composite elements that have failure indices requested.

NOFISR - controls the printout of the composite failure indices and strength ratios. When 'on', the failure indices and strength ratios will not be printed.

CNTASET - used to perform a static condensation on the contact degrees-of-freedom for linear contact (NX Nastran only). Contact iterations are then performed using the reduced matrix, which should decrease solve time.

BAILOUT - used to allow under constrained models continue to run if any mechanism (free motion) is detected.

AUTOMPC - when enabled, writes PARAM, AUTOMPC, YES (NX Nastran only). Allows the software to automatically select the m-set degrees-of-freedom instead of using the m-set values specified by MPC or on rigid elements (RBE1, RBE2, RBE3, etc). There are some caveats, see NX Nastran Quick Reference Guide.

DDRMM off- when enabled, writes PARAM, DDRMM,-1, which forces calculation of complete g-set solution vectors be the mode displacement method in modal transient and frequency response solutions. When disabled, which is the default, writes nothing, which means the matrix method of data recovery will be used.

MODACC - when enabled, also enables *DDRMM off*, then writes PARAM, MODACC,0. This specifies the mode acceleration method for data recovery in dynamic analysis. When off, nothing is written.

RESVEC and ***RESVINER*** - allows you to write two forms of the RESVEC PARAM. PARAM,RESVEC,NO which augments static shapes due to applied loads; or PARAM, RESVEC,YES which computes residual vectors for applied loads and unit loads (with specified USETi, U6 entries at the desired DOF). When RESVEC is not checked, the PARAM,RESVEC entry will not be written at all, which is required for some types of analysis. To augment static shapes due to inertial loads (unit acceleration of mass), check RESVINER to write PARAM,RESVINER,YES.

SECOMB - used to control if output will be combined for a superelement analysis, has a number of caveats.

APLHA1 and ***ALPHA2*** - ALPHA1 is the complex scale factor applied to the mass matrix and ALPHA2 to the stiffness matrix. Used in frequency and transient response analysis, if PARAM,ALPHA1 and/or ALPHA2 are not equal to complex zero, then Rayleigh's damping is added to the viscous damping.

Format

These options determine the format that will be used to write your Bulk Data commands. By default, FEMAP uses small field format (8 character fields). If you want extra precision for some or all of your model, you can choose one of the large field formats (16 character fields). The large field formats obviously produce a large file that is harder to read. You should not choose that format unless it is necessary. The "limited" large field formats allow you to selectively write large field formats for certain entities and small field format for others. FEMAP does not write free field format.

Translator Options

All Plates as QUADR/TRIAR - Nastran supports two plate formulations:

- The CQUADR and CTRIAR elements have rotational stiffness in the direction normal to the plane of the element.
- By default, the CQUAD4 and CTRIA3 elements will be written. These do not have any rotational stiffness in the normal direction.

Skip Beam/Bar Cross Sections - Nastran can use PBEAM entries or PBEAML entries to define beam properties. You can create both PBEAMs and PBEAMLs in FEMAP using the *Model, Property...* command.

- FEMAP computes values for a "Standard Beam" from the cross-section data supplied and enters the values into the appropriate fields on the *Define Property - BEAM Element Type* dialog box. When a Nastran input deck is exported, FEMAP creates a PBEAM entry for each "standard beam" property defined in the model. Nastran then uses the PBEAM data as it would any other property data to analyze your structure
- When FEMAP creates a "NASTRAN Beam", the cross-section data supplied is also used to compute values and enters them into the appropriate fields on the *Define Property - BEAM Element Type* dialog box. Upon export to Nastran, FEMAP instead creates PBEAML entries for each "NASTRAN Beam" in the model.

PBEAML entries contain cross-section dimension data corresponding to a specific PBEAML Type specified on each PBEAML entry. Nastran uses this cross-sectional data and PBEAML type internally to analyze the structure.

Sometimes you may want to only export PBEAM entries out of FEMAP for analysis purposes. By choosing *Skip Beam/Bar Cross Sections*, FEMAP will use the computed property values from the *Define Property - BEAM Element Type* dialog box and only create PBEAM entries in your Nastran input file, regardless of how the beams were defined.

An example of when this option would be used, would be if you have a model created with “NASTRAN Beams”, which needs to be run by a version of Nastran that does not support PBEAML entries.

Rigid Element Thermal Expansion - Both NX Nastran (version 5.1 and above) and MSC/MD Nastran (version 2005 and above) support a Coefficient of Thermal Expansion for Rigid Elements. This box must be CHECKED in order for the CTE to be written out to NX and MSC/MD Nastran.

Note: In FEMAP, the Rigid element CTE is defined by using the “Coefficient” field in the *Thermal Expansion* portion of the *Define RIGID Element* dialog box.

Gaps as Contact - In NX Nastran only, checking this option will allow gap elements to be treated as linear contact elements during a Linear Static analysis (SOL 101). This option creates a BCSET in the case control section of the NX Nastran file, which is what tells NX Nastran to use the gaps as linear contact elements.

Note: If you have linear contact defined elsewhere in your model using the entities created on the *Connect* menu, there is no need to turn this option on for gaps to work as linear contact elements, as a BCSET is already being created in the case control.

Dynamic Loads using LOADSET/LSEQ - Writes out loads for dynamic analysis using LSEQ method. This was the method used in Nastran before direct application of dynamic loads became available.

Write All Static Loads/BCs Sets - When this option is on, ALL loads and constraint sets will be written to the Nastran input file for Linear Static Analysis. This essentially forces FEMAP to write out Nastran input files for SOL 101 the way it has in all versions before FEMAP 10.1.

Manual Control

- If the *Skip Standard Bulk Data* switch is off, the software writes standard Bulk Data. If this switch is on, the software writes ONLY the contents of *Start Text* or *End Text* that you enter.
- Pick *Start Text* to add text to the beginning of the Bulk Data section, after the BEGIN BULK command.

Note: When using the “bundled” version of NX Nastran which comes with FEMAP with NX Nastran, only Bulk Data entries AFTER the BEGIN BULK entry may be in this section. The BEGIN BULK entry itself is a Case Control command and MUST NOT be in this section, even via an INCLUDE file. Otherwise, the “Checksum”, which is used for licensing, will not be calculated correctly and the analysis will not run.

- Pick *End Text* to add text to the end of the Bulk Data section after the END DATA command.

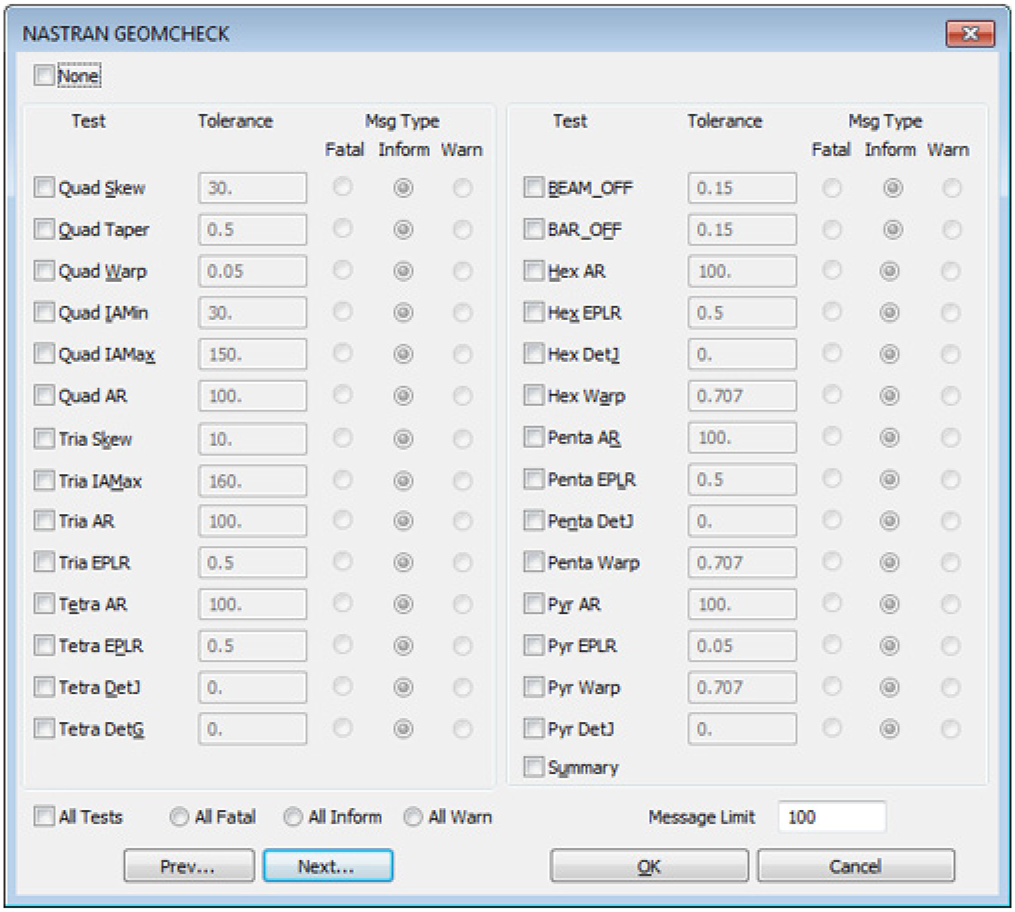
Note: Nastran INCLUDE Statements pointing to “Include files” can be used in NX Nastran for FEMAP

8.7.1.4 NASTRAN GEOMCHECK

The *NASTRAN GEOMCHECK* dialog box allows you to have Nastran perform element quality checks using the *Tolerance* value specified for each selected type of element. These entries are written out to the Executive Control Section of the Nastran file.

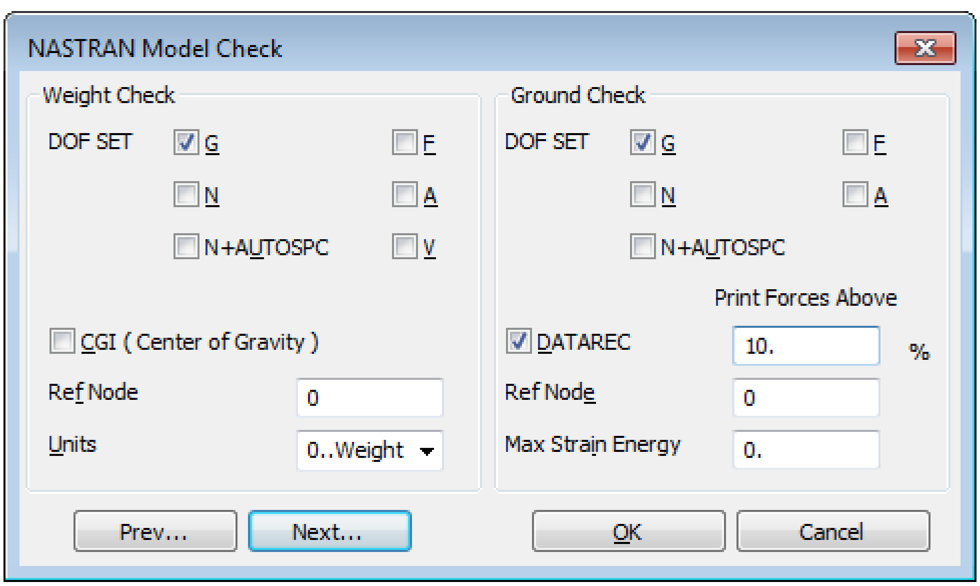
For each test a tolerance is chosen and the action to be performed if the tolerance is surpassed. If *Fatal* is chosen then the analysis will stop with a fatal message, if *Inform* is selected then Nastran will inform you of the entities that did not pass the quality check. *Warn* will instruct Nastran to issue warning messages when the tolerance is surpassed. Message Limit sets the maximum number of messages that will be issued.

Choosing *None* in the upper left corner of the *NASTRAN GEOMCHECK* dialog box will write GEOMCHECK, NONE to the Executive Control Section.



8.7.1.5 NASTRAN Model Check

Opens the *NASTRAN Model Check* dialog box which allows you to choose a number of different options to have Nastran perform a “ground check” or a “weight check”.



The *Weight Check* section allows you to define the WEIGHTCHECK case control card for the Rigid Body Mass Reduction Check. Choose the degree-of-freedom set and a reference grid point. If no grid point is selected then the origin of the basic coordinate system is used.

- *CGI (Center of Gravity)* option requests output at the center of gravity

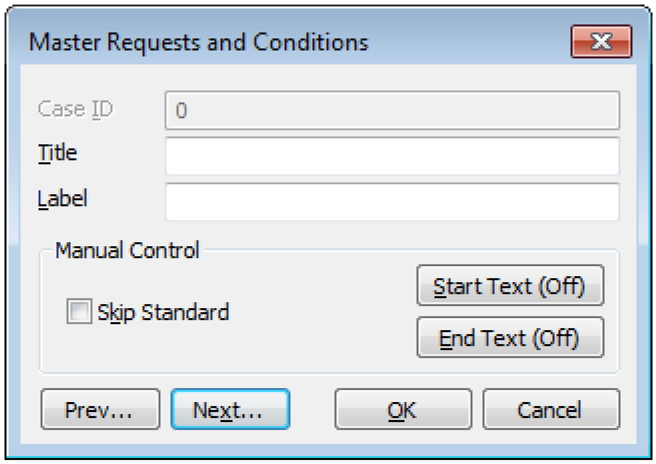
- *Units* chooses weight or mass units

Ground Check defines the options for the GROUNDCHECK case control command. Pick a degree-of-freedom set and the set the options you want.

- *DATAREC* will request the recovery of grounding forces that are above the tolerance set.
- *Ref Node* for the calculation of the rigid body motion, while *Max Strain Energy* represents the maximum value which passes the check.

8.7.1.6 Master Requests and Conditions

Master Requests and Conditions define the master output requests and boundary conditions (loads and constraints) for your analysis. On the *Master Requests and Conditions* dialog box, you can enter a *Title,* a *Label*, and *Manual Control* options. Once you have entered this data, pick *Next* to set up the boundary conditions.



The *Manual Control* options include:

- *Skip Standard*: If this switch is on, the interface does not write text to the input file. If *Start* and *End Text* have been defined, they will still be written to the Case Control section of the input file

Note: When skipping the standard Case Control and using the “bundled” version of NX Nastran which comes with FEMAP with NX Nastran, be sure to add the BEGIN BULK entry as the final line in the *End Text* dialog box (or *Start Text* dialog box, if not using *End Text* or any Subcases) of the *Master Requests and Conditions*. If using INCLUDE files, the BEGIN BULK entry MUST be in the INCLUDE file referenced by the *Master Requests and Condition*s. Otherwise, the “Checksum”, which is used for licensing, will not be calculated correctly and the analysis will not run to completion.

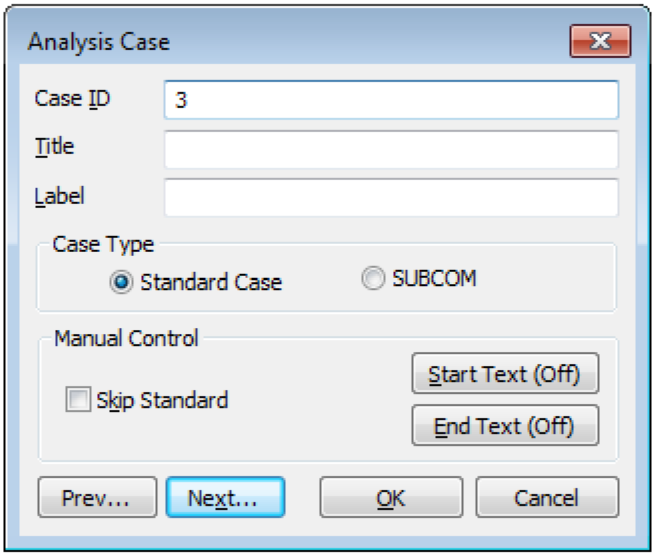
- *Start Text*: Pick this option to add text to the beginning of the Case Control section of the input file.
- *End Text*: Pick this option to add text to the end of the Case Control section of the input file.

Hint: One easy way to create cases is to use the *MultiSet* button on the *Analysis Set Manager. Multi-Set* creates one case for each combination of loads and constraints.

Cases

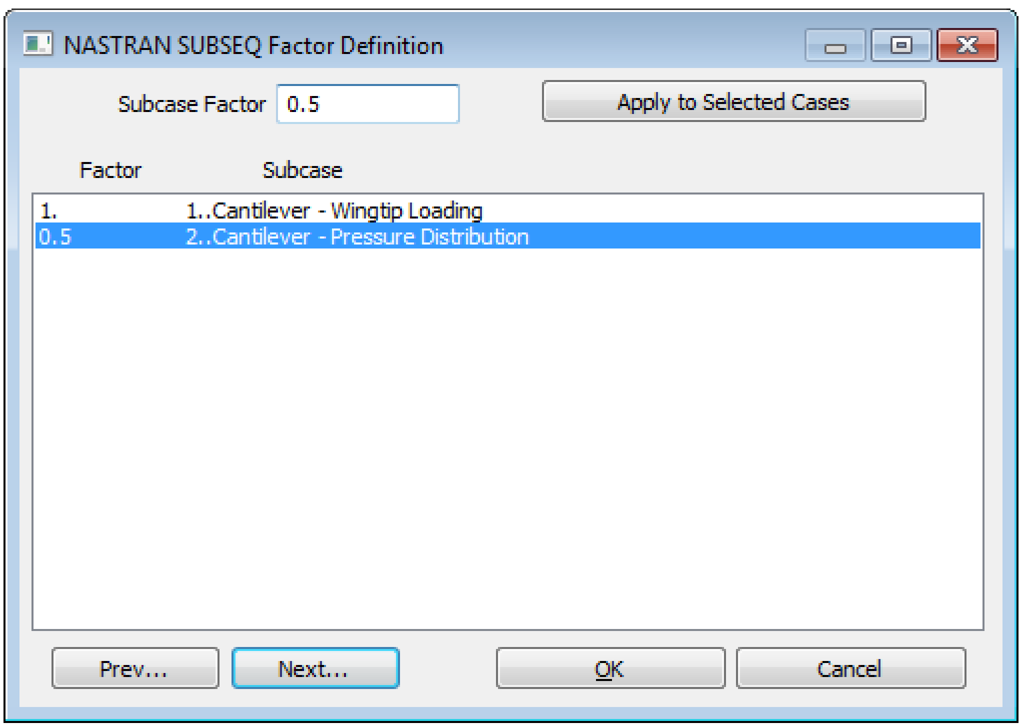
In addition, you can define cases which let you perform multiple analyses with different load and/or constraint sets. You can also specify output requests for each case. The analysis program will generate one output set for each case.

Use the *Analysis Case* dialog box to enter a *Case ID*, *Title*, and *Label* for a case. For Linear Static analysis in Nastran, you have the choice of creating a “Standard Case” or a “SUBCOM”, which is a combination of other Subcases defined in your model.



Sometimes in Nastran each case may require different "manual control" text. Once you have entered this data, pick *Next* to continue setting up the analysis. (The master requests and conditions provide the defaults for the cases.)

When "SUBCOM" is chosen as the *Case Type*, clicking *Next* will open the *SUBSEQ Factor Definition* dialog box.



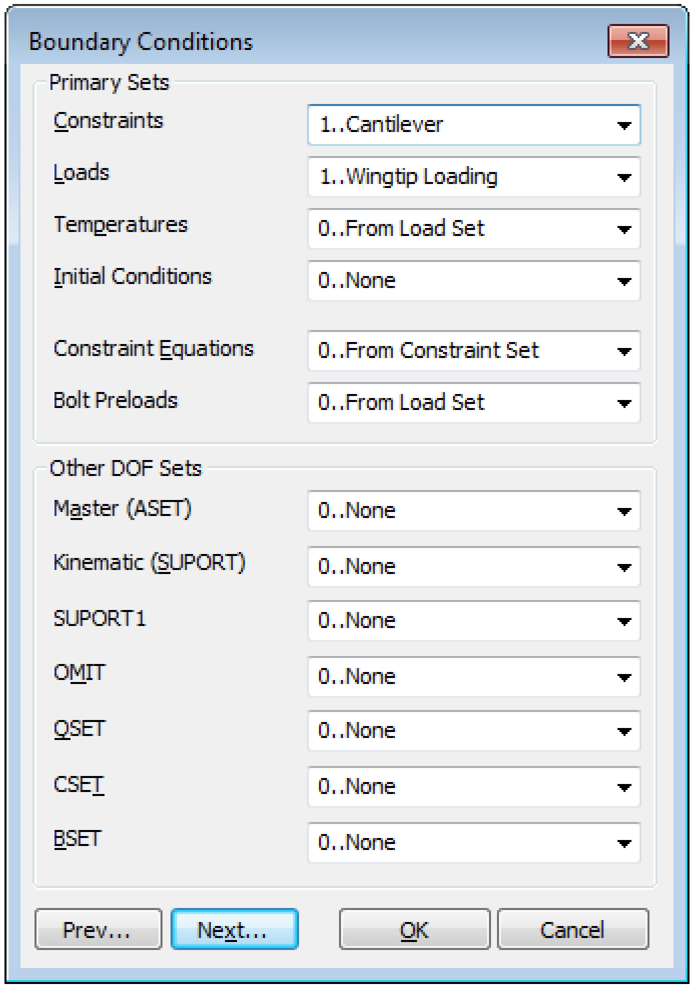
In this dialog box, all existing subcases will be listed. You can now highlight one or more of the subcases and enter a *Subcase Factor*. Clicking the *Apply to Selected Subcases* button will update the "Factor" in the list window of this dialog box. The "Factor" is a simple scale factor which will be used by Nastran to combine the selected subcases in the prescribed manner.

Note: Once your "SUBCOM" has been defined, clicking *Next* will then take you to the *Boundary Conditions* dialog box where you can set a new Initial Temperature using the Load Set (all other will be ignored).

For example, you may want to run a subcase that uses half of "Subcase A" and 2 X "Subcase B". For this you would highlight "Subcase A" and enter a *Subcase Factor* of "0.5", then click the *Apply to Selected Subcases* button. Next, you highlight "Subcase B" and enter a *Subcase Factor* of "2", then click the *Apply to Selected Subcases* button.

8.7.1.7 Boundary Conditions

The *Boundary Conditions* dialog box lets you select the loads and constraints to apply to your analysis. You can apply boundary conditions as both master boundary conditions or in cases.



Primary Sets

Depending on your analysis type, you can select constraints and loads.

- *Constraints*: pick a constraint set for your model.
- *Loads*: pick a load set for your model.
- *Temperatures*: pick a load set containing temperature loads for the model. Typically, this is used to apply temperatures to a structural model and will create a TEMP(LOAD) entry in Case Control.

Note: If a Nastran LOAD Combination is selected using the *Loads* drop-down, then it is a good idea to place any temperature loads in a different load set and select it using the *Temperatures* drop down.

- *Initial Conditions:* pick a load set to use for initial conditions.

Note: Although it is often not needed, *Initial Conditions* can be set up for thermal strain problems in Linear Static Analysis. The only type of initial conditions supported by Nastran SOL 101 are thermal. Make sure if you are selecting an *Initial Condition* Load Set for Linear Static that you have either defined individual nodal temperatures, set the Default Temperature as a Body Load, or a combination of both. The TEMP (INIT) card will be written in the Case Control section of the Nastran input file and the temperatures set in the initial conditions Load Set will override the Reference Temperature (TREF) field on MATi entries.

- *Constraint equations:* pick a constraint set to define constraint equations. If you choose *From Constraint Set*, FEMAP will look for constraint equations in the same set as your nodal constraints. This is a convenient way to manage most models.
- *Bolt Preloads:* pick a specific load set for bolt preloads in your model. By default, the bolt preloads will be specified in the “Primary Load Set” and will run fine in NX Nastran. This option simply allows you to separate the regular loads from the bolt preloads so you can easily run both a “with preloads” and “without preloads” analysis without having to actually modify the “Primary Load Set”.

Note: Having your Bolt Preloads in separate Load Set(s) is also very helpful if you have to run the same model with multiple “bolt preload conditions”. Simply create a Load Set for each “bolt preload condition” and you can switch from one to another using this drop-down list.

For heat transfer analyses, you will notice that constraint sets are not used. Rather, loads and constraints are both selected from a load set. FEMAP translates nodal temperatures, in the same set as the other thermal loads, as thermal constraints (boundary conditions).

Note: If your analysis requires multiple load or constraint sets, you must create cases.

Other DOF Sets

You can select constraint sets to use as various types of DOF sets.

Typically, for static analysis you will not want to choose any of these sets. For some types of modal analysis, you can choose an ASET to reduce the number of analysis degrees of freedom (Guyan Reduction). The other sets are used less frequently.

For “Free-Free” Modal Analysis, it is often required to use the Kinematic (SUPORT) set to create “fictitious supports” which allow Nastran to run with no actual constraints defined.

Specifying the SUPORT1 DOF set will create a SUPORT1 case control entry along with the appropriate number of SUPORT1 bulk data entries to represent a “fictitious support” set. This can be useful because the SUPORT1 can be specified in a subcase, while the SUPORT DOF set will be used in all subcases.

OMIT, QSET, CSET, and BSET use constraint sets to write out the appropriate OMIT, QSET, CSET, and BSET bulk data entries.

Contact/Glue Sets (MSC Nastran Only)

The upper portion of this section, only available for MSC Nastran, allows you to select which connectors in the model will be written to the BCTABLE entry in the MSC Nastran input file and if they will be used:

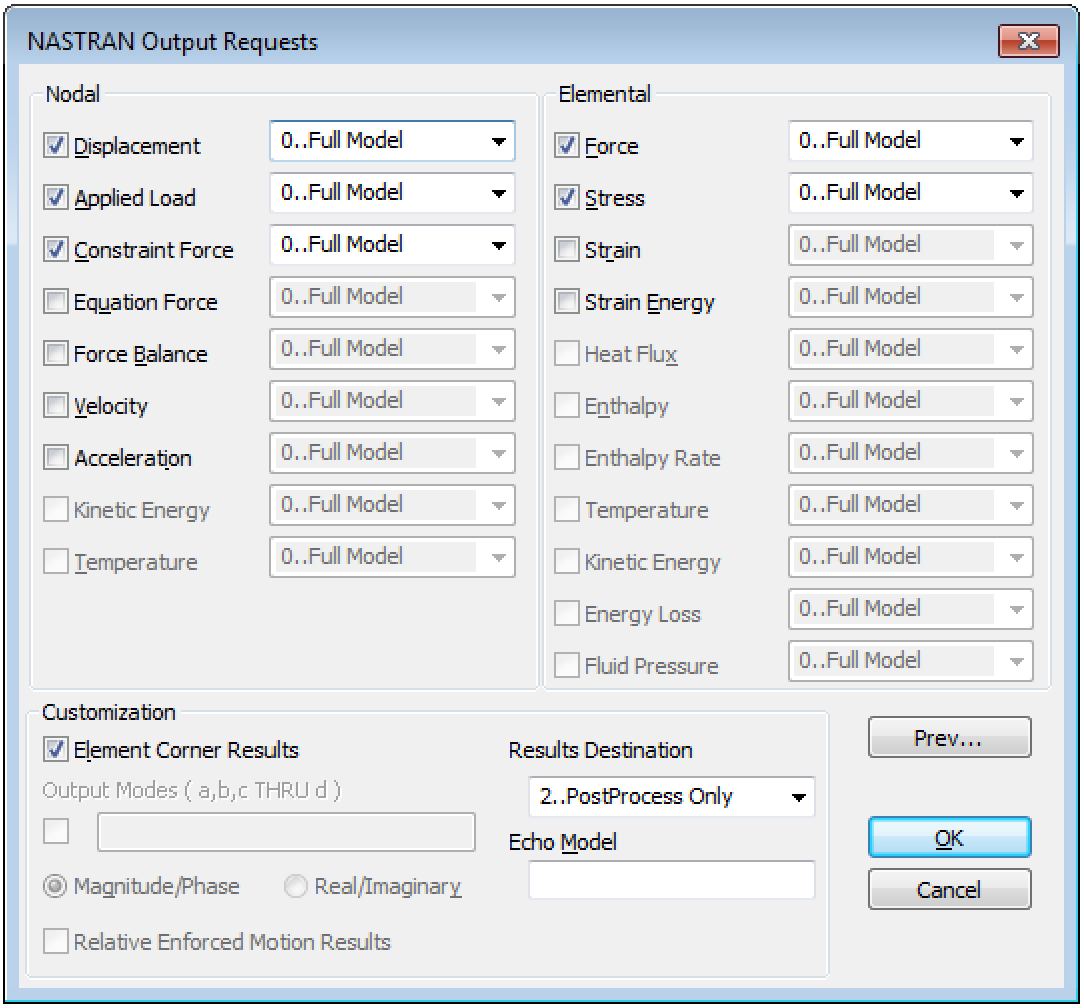
- *All Connectors* (default) - simply writes all connectors currently enabled in the model to the input file.
- *Connection Group* - allows you to select a group containing any number of connectors from the drop-down control, then writes out only those connectors. The *Quick Group* icon button can be used to quickly create a new group, then you can choose *Edit Group* to select connectors in the model to add to the selected group.
- *None* - writes all enabled connectors out to the input file, but does not write BCONTACT to Case Control.

The *BCONTACT Options* portion controls how the BCONTACT entry is written in Case Control:

- *Default* - for *All Connectors*, writes BCONTACT=107 and sets ID of BCTABLE entry t to 107, while for *Connection Group*, writes BCONTACT=108 and sets ID of BCTABLE entry t to 108. Can be used for either Linear or Glued contact.
- *Initial Contact* - writes BCONTACT=0 and sets ID of the BCTABLE entry to 0. Only for Linear contact.
- *All Body* - writes BCONTACT=ALLBODY and the BCTABLE is not written at all. Only for Linear contact.

8.7.1.8 NASTRAN Output Requests

Use the *Output Requests* dialog box to identify the types of output that you want from the analysis. The type of output that you can request will depend on the analysis program and analysis type.



You can define output requests as both master output requests or as part of a case.

Some analysis types such as Normal Modes allow you to define additional options. The options available depend on the analysis type you have selected.

For frequency analyses, you can recover output in either *Magnitude/Phase* or *Real/Imaginary* format.

The *Relative Enforced Motion Results* can be used recover relative results from an enforced motion analysis.

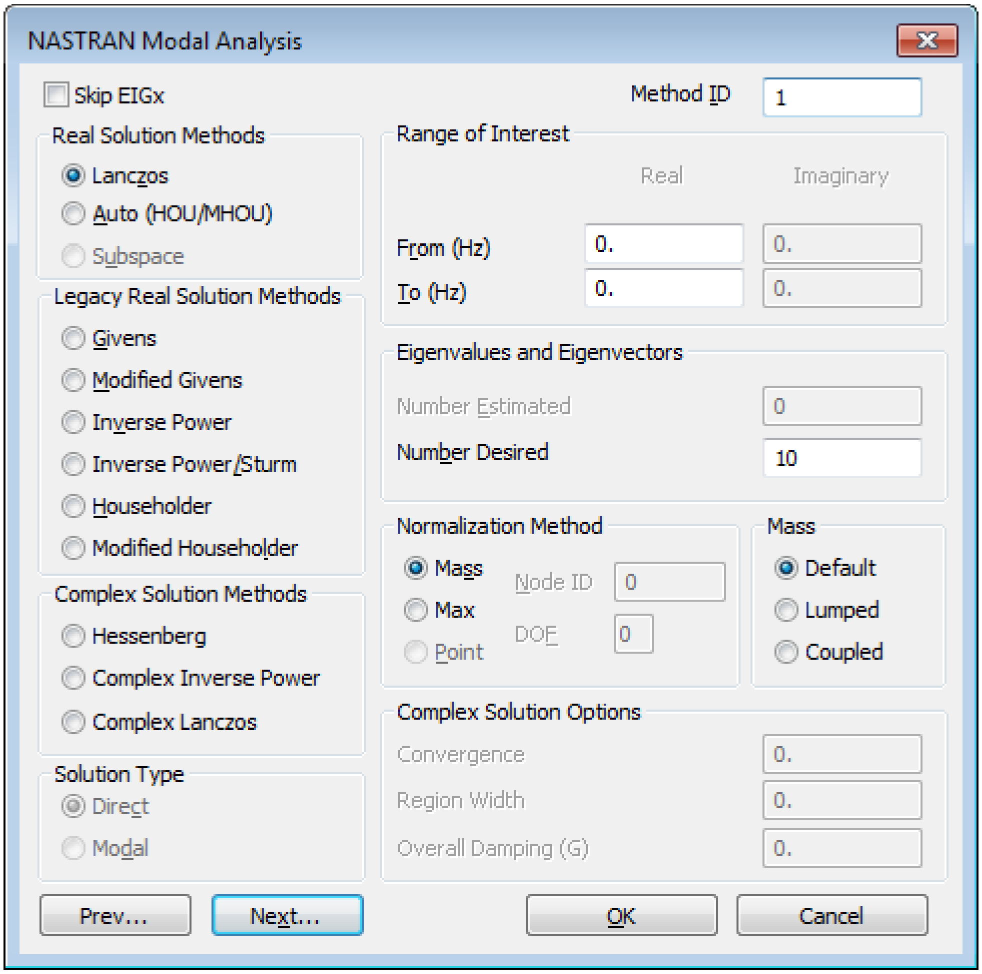
Customization also allows you to select a results destination file (PostProcess only = *.op2, Print Only = *.f06, XDB = *.xdb, etc.).

Note: When you select "3..Print and PostProcess" as the *Results Destination*, you are sending the results to both the .f06 and the .op2 file. Normally, you would not want to do this, but the option is there to complete all the possible combinations for requesting output. When FEMAP runs NX Nastran, it automatically reads the results (you can change this with a preference: *File, Preferences,* then click *Interfaces*, then uncheck box "Automatically Load Results"), but it does it by first reading the .f06 file. FEMAP reads the .f06 file first to attain any error, warning, or information messages that might have occurred during the analysis.

If you are requesting grid point force data to create Freebody plots in FEMAP, you must choose the "2..PostProcess Only" option or the "6..XBD" option, as the grid point force data is not in the .f06 file.

8.7.1.9 NASTRAN Modal Analysis

The *NASTRAN Modal Analysis* dialog box lets you enter options for Normal Modes, Random, or Buckling Analysis. These parameters are used to define the EIGR (EIGRL, EIGC, EIGB) command that controls modal analysis.



A similar box appears for “3..Transient Dynamic/Time History” and “4..Frequency/Harmonic Response” analysis, except you have an option to select *Direct* or *Modal* solution type (also available when using one of the *Complex Solution Methods*).

Skip EIGx

The *Skip EIGx* option allows you to have FEMAP NOT write any *“EIGx”* (EIGC, EIGR, EIGRL) entries into the Nastran input file (*.dat or *.bdf file). If FEMAP does not support a particular option on any of the *EIGx* entries which you might want to use, you can “skip” writing an *EIGx* entry out during export and instead use the “Start Text” button in the *Manual Control* section in the *NASTRAN Bulk Data Options* to “manually” enter an *EIGx* entry.

Method ID

The *Method ID* specifies the ID of the EIGR command. It is also used in the Case Control section on the METHOD command to select the *EIGR* command.

Real Solution Methods and Legacy Real Solution Methods

The options in these sections allow you to choose the method that will be used for real modal extraction. The options will be designated using the *EIGR*, *EIGRL,* or *EXTRACTMETHOD* (NEi Nastran only) entry in the Nastran input file. Depending on which solution method you choose, the various other options can be used in different combinations. are “legacy methods”. While the options in *Legacy Real Solution Methods* section are still valid methods of eigenvalue extraction, it may be to your advantage to use one of the options in the *Real Solution Method* section. Refer to your Nastran documentation for more information regarding which method will be best for your model.

Complex Solution Methods

The options in this section allow you to choose the method that will be used for complex modal extraction. The options will be designated using the *EIGC* entry in the Nastran input file. Depending on which solution method you choose, the various other options can be used in different combinations.

Note: By selecting a *Complex Solution Method*, the *Imaginary* fields in the *Range of Interest* section will become available, along with the *Direct* and *Modal* options in *Solution Type*. When using a *Complex Solution Method* for NEi Nastran, the *Solution Type* MUST be set to *Modal*.

Solution Type

The *Solution Type* section is used to specify what solution type should be used when creating analysis sets for “3..Transient Dynamic/Time History” or “4..Frequency/Harmonic Response” Analysis Types. *Direct* is used to create Direct Transient or Direct Frequency, while *Modal* creates a Modal Transient or Modal Frequency analysis.

If you pick the Modal solution type, you can also enter modal participation information on another dialog box. (See Section 8.7.1.12, "NASTRAN XY Output for Modal Analysis".)

Range of Interest

These options select the range of frequencies where eigenvectors will be computed. There are fields to specify a range for both Real (Normal Modes) and Imaginary (Complex Modes). Specifying a value of 0.0 in the *From* field will write a “blank” field, which may or may not be desirable, while entering a value of 1.0E-15 will cause Nastran to use a value which is essentially 0.0.

Eigenvalues and Eigenvectors

The estimated number of roots is only used for the *Inverse Power* solution method (where it is required). The number of desired vectors is typically an alternative to the frequency range. Instead of specifying frequencies, you can choose to recover a number of eigenvectors with the lowest frequencies. Entering a value below 0 in the *Number Desired* field will write a “blank” field, which may or may not be desirable.

Normalization Method

These options choose the method for eigenvector normalization. *Mass* normalizes to the unit value of the generalized mass. *Max* normalizes to the largest component of mass in the analysis set, and *Point* normalizes to the mass at a specific nodal degree of freedom. If you choose point normalization, you must also specify a node ID and degree of freedom.

Mass

Allows you to designate if the mass matrices for elements with coupled mass capability (i.e., CBAR, CBEAM, CQUAD4, CHEXA, CPENTA, CQUAD8, CROD, CTETRA, CTRIA3, CTRIA6, CTRIAX6, and CTUBE) should be “Coupled” or remain “Lumped” (default for most analyses) by writing the PARAM, COUPMASS entry in the Nastran input file. When “Coupled” is selected, both structural and non-structural mass are taken into account for the aforementioned elements.

Complex Solution Options

Convergence is a Convergence Criteria (E field in the EIGC entry) which can be entered when using any of the *Complex Solution Methods*. The default values for each method is different and are as follows:

Hessenberg = 10^{-15}, *Complex Inverse Power* = 10^{-4}, *Complex Lanczos* is machine dependent.

Region Width is only available when using the *Complex Inverse Power* method and is the width of the j-th Search Region which by default is 1.0 and is entered in the Lj field on the EIGC entry. *Overall Damping (G)* - sets the “Overall Structural Damping” for complex modal analysis using the PARAM, G.

Note: In most cases, a damper element or “G” will be used to provide “Damping” for a Complex Modes analysis. The PVISC and CVISC entries can be created in FEMAP using the *Spring/Damper* Property (set to “Other (NASTRAN CROD/CVISC)”) and Element.

Note: When displaying results for “Complex Modal Analysis” the default Titles displayed in the graphics window will contain both the *Frequency* and the *Damping Coefficient*.

Special Note about Pre-stiffened Modal Analysis.

You will notice that during a Normal Modes analysis, that you cannot choose a Load Set in the *Boundary Conditions* dialog box of the Master Analysis Case. You can run a pre-stiffened modal analysis in FEMAP by adding a loading condition to your model, but it does require a subcase to be created in order for it to work properly. The exact process is to go to the bottom of the "tree structure" in the Analysis Set Manager and highlight the "No Case Defined" branch of the "tree". Now click the *New* button and FEMAP will prompt you to create a subcase. Click *Next* and you can now add a loading condition to the model using the *Loads* pull-down menu (you will need to have a load set for this to work properly). You will notice an additional SUBCASE has been added to the Case Control section and a STATSUB entry can be found there which will prompt Nastran to run a Pre-stiffened Modal Analysis. Now run the analysis and your results will reflect pre-stiffened modal analysis. A pre-stiffened modal analysis is needed when using Linear Contact with NX Nastran SOL 103.

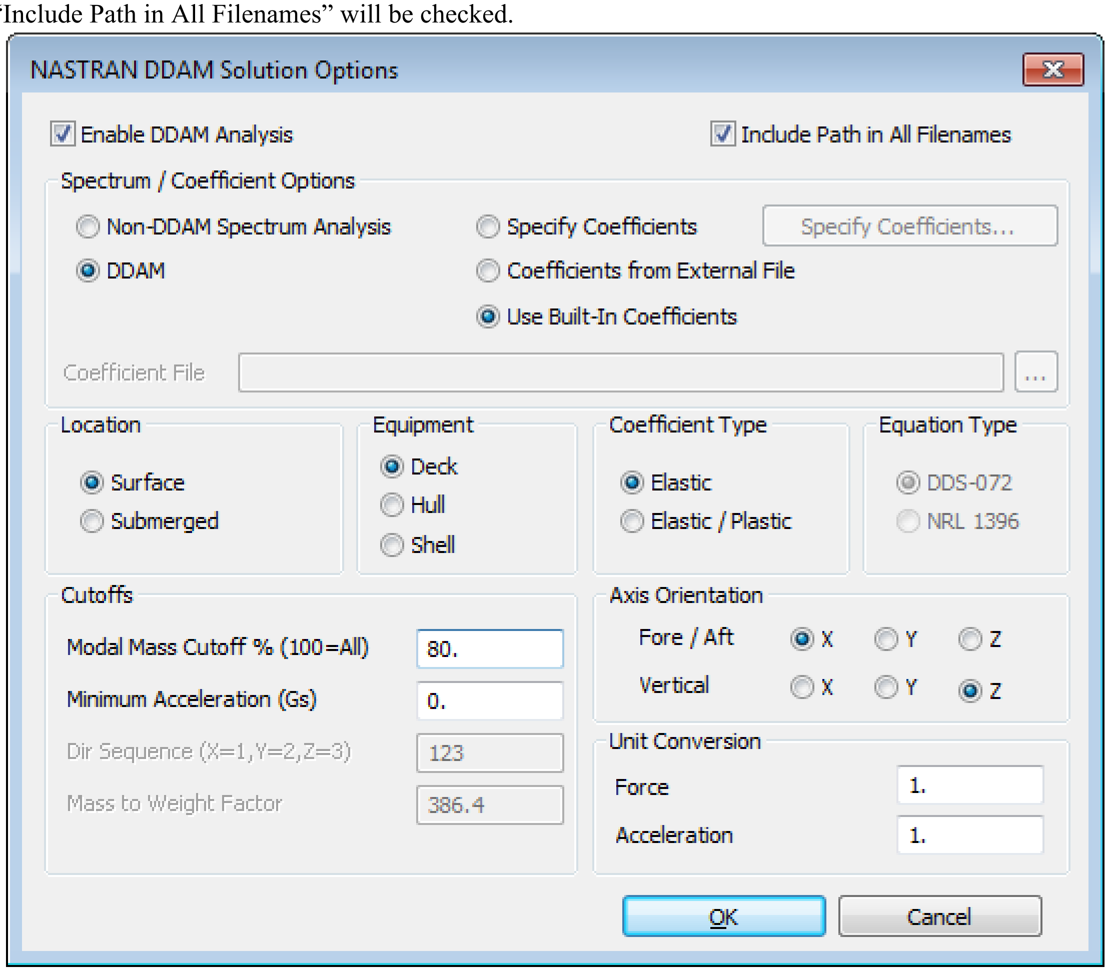
Note: For performing "Stiffened Modes in Nonlinear Analysis" (SOL 106), you will need to "expand" the *Options* portion of the tree in the *Analysis Set Manager*, highlight *NASTRAN Stiffened Model* from the list, then click the *Edit* button. The *NASTRAN Modal Analysis* dialog box will appear with the "Enable Stress Stiffening" option already checked and other modal options can be specified

8.7.1.10 NASTRAN DDAM Analysis (Modal Analysis Only)

The Dynamic Design Analysis Method (DDAM) is a list of procedures to determine the modal shock response of on-board ship equipment due to underwater explosions. In NX Nastran and MSC/MD Nastran, DDAM analysis has been implemented as a single solution sequence, SOL 187. For NEi/Nastran, the available options will be written out to a "DDAMDAT" entry (instead of a NAVSHOCK file) and used appropriately.

Note: To access the DDAM options in FEMAP, you will need to "expand" the *Options* portion of the tree of an Analysis Set (Modal Analysis only) in the *Analysis Set Manager*, highlight *DDAM* from the list, then click the *Edit* button. The *NASTRAN DDAM Solutions Options* dialog box will appear.

You must check the "Enable DDAM Analysis" box for FEMAP to use any DDAM options. By default, the "Include Path in All Filenames" will be checked.



This solution sequence has 3 separate phases for SOL 187 which all occur automatically from a single Nastran job submittal, and can be set-up completely from within FEMAP:

• Phase 1: A modal analysis (SOL 187) runs to calculate the natural frequencies. Then the participation factors and modal effective weights are calculated for each mode. The modes, participation factors, and modal effective weights are written to an ASCII OUTPUT4 file.

• Phase 2: The Naval Shock Analysis (NAVSHOCK) FORTRAN program is automatically invoked to compute the modal shock responses. NAVSHOCK uses the following files as input:

The OUTPUT4 file created in phase 1 by NX Nastran.

- A required, user-created DDAM Control file storing various runtime options
- An optional, user-created DDAM Coefficient file containing the weighting factors used for the response calculations, the directional scaling factors, as well as the modal mass cutoff value. This file must be listed in the DDAM Control file for NAVSHOCK to use it.
- An optional, user-created Shock Spectra file which defines the input shock spectrum as data pairs of frequency and displacement, velocity, acceleration. This file must also be listed in the DDAM Control file for NAVSHOCK to use it.

FEMAP creates all of the required files and other controls based on what is entered in the *NASTRAN DDAM Solution Options* dialog box

• Phase 3: The modal shock responses created in phase 2 are read by Nastran, and results are recovered and output for post-processing.

Here is a listing of how the “lines and items” of the NAVSHOCK control file correspond to various sections and options of the *NASTRAN DDAM Solution Options* dialog box. The names for the options are often very similar or identical to how they are described in the NX Nastran documentation. The FEMAP option in the *NASTRAN DDAM Solution Options* dialog box is in parentheses after the appropriate Line or Item

Spectrum/Coefficient Options and Equation Type

First Line - spectrum control

First Item - Coefficients from File or from compiled source (“Second Item” in MSC/MD Nastran)

T = coefficients from external file (*Coefficients from External File* Radio Button)

F = use built-in coefficients (*Use Built-In Coefficients* Radio Button)

Specify Coefficients Radio Button - allows you to click the *Specify Coefficients* button and enter a the coefficients in this dialog box. FEMAP will then write the corresponding “Coefficient File” when the analysis is run.

Second item - DDAM or general spectrum run flag (“First Item” in MSC/MD Nastran)

T = General non-DDAM spectrum run (*Non-DDAM Spectrum Analysis*)

F = DDAM (*DDAM*)

Third item - Equation Format (MSC/MD Nastran Only)

T = DDS-072 style equations (*DDS-072* Radio Button in *Equation Type* section)

F = NRL 1396 style equations (*NRL 1396* Radio Button in *Equation Type* section)

Second Line - file name (if needed - If neither *First Item* or *Second Item* are T, line is not needed)

If 1st item on line 1 is T, Name of coefficient file (*Coefficient File* field when *DDAM* is chosen)

If 2nd item on line 1 is T, Name spectrum file (*Spectrum File* field when *Non-DDAM Spectrum Analysis* is chosen)

Note: You can use the *“...”* button to browse to the appropriate file.

Location, Equipment, and Coefficient Type

Third Line - location flags

First Item - Surface or Submarine (*“Location”* Section)

1=Surface (*Surface* Radio Button in *Location* Section)

2=Submarine (*Submerged* Radio Button in *Location* Section)

Second Item - equipment location (*"Equipment"* Section)

1=Deck (*Deck* Radio Button in *Equipment* Section)

2=Hull (*Hull* Radio Button in *Equipment* Section)

3=Shell (*Shell* Radio Button in *Equipment* Section)

Third Item - coefficient type (*"Coefficient Type"* Section)

1=Elastic (*Elastic* Radio Button in *Coefficient Type* Section)

2=Elastic/Plastic (*Elastic/Plastic* Radio Button in *Coefficient Type* Section)

Cutoffs, Axis Orientation, and Unit Conversion

4th Line - Weight cutoff percentage - must be from 0. To 100. (*Modal Mass Cutoff %* in "*Cutoffs"* Section)

5th Line - Axis Orientation (these options write "6th Line" in NAVSHOCK for use with MSC/MD Nastran)

First Item - F/A axis, X, Y, or Z (*Fore/Aft X, Y,* and *Z* Radio buttons in the *Axis Orientation* Section)

Second Item - Vertical Axis X, Y, or Z (*Vertical X, Y,* and *Z* Radio buttons in the *Axis Orientation* Section.

Minimum Acceleration (Gs) - Minimum G level to use (in Gs). Writes the "5th Line" in NAVSHOCK for use with MSC/MD Nastran and the "9th Line" for NX Nastran.

Unit Conversion - *Force* written to "10th Line", *Acceleration* written to "11th Line" for NX Nastran.

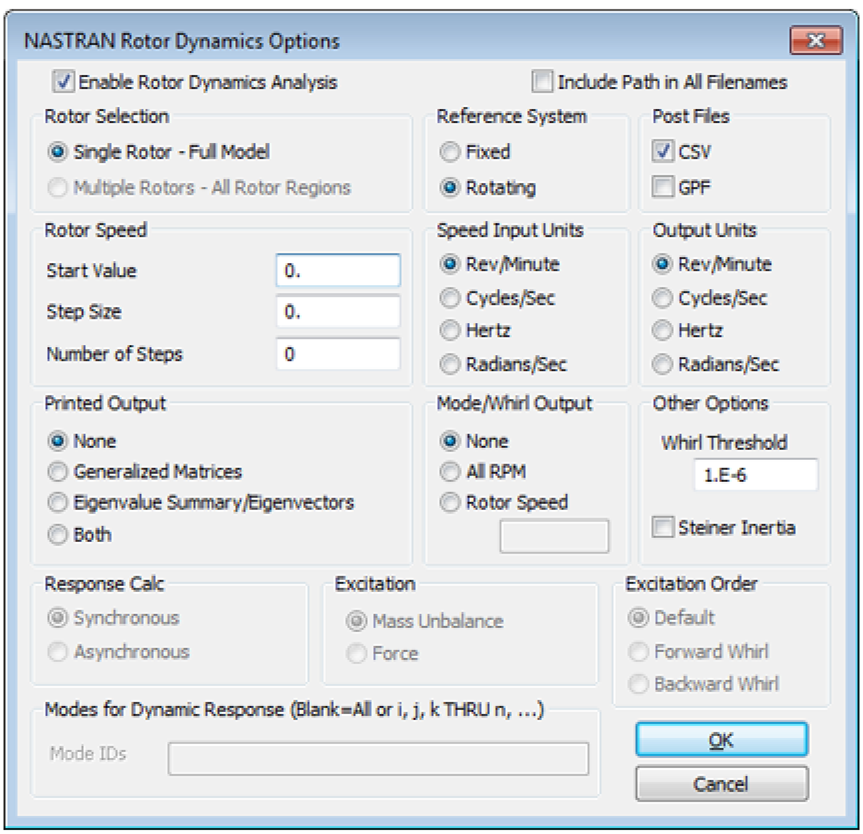
Dir Sequence (X=1,Y=2,Z=3) and *Mass to Weight Factor* are for NEi/Nastran only.

8.7.1.11 NX Nastran Rotor Dynamics (SOL 110 and 111 Only)

Rotor Dynamics are available in Complex Modal Analysis (SOL 110) and Modal Frequency Response (SOL 111).

Note: To access the NX Nastran Rotor Dynamic options in FEMAP, you will need to "expand" the *Options* portion of the tree of an appropriate Analysis Set in the *Analysis Set Manager*, highlight *NASTRAN Rotor Dynamics* from the list, then click the *Edit* button. The *NASTRAN Rotor Dynamics Options* dialog box will appear.

You must check the "Enable Rotor Dynamic Analysis" box for FEMAP to use any Rotor Dynamics options.



Individual “Rotors” can be created in FEMAP using the *Connect, Rotor Region*. See Section 4.4.9, "Connect, Rotor Region..." in the FEMAP Commands Manual.

Rotor Selection

By default, FEMAP will use the *Single Rotor - Full Model* option. When this is set, only the first two lines of the ROTORD entry will be written. When *Multiple Rotors - All Rotor Regions* is chosen, each Rotor Region in your model will write a separate continuation line for the ROTORD containing all of the Rotor information (RIDi, RSETi, RCORDi, W3_i, W4_i, and RFORCEi).

Post Files

Checking *CSV* and/or *GPF* will write the appropriate “ASSIGN OUTPUT4” statement(s) to the NX Nastran input file which will generate the rotor.csv and/or rotor.gpf files.

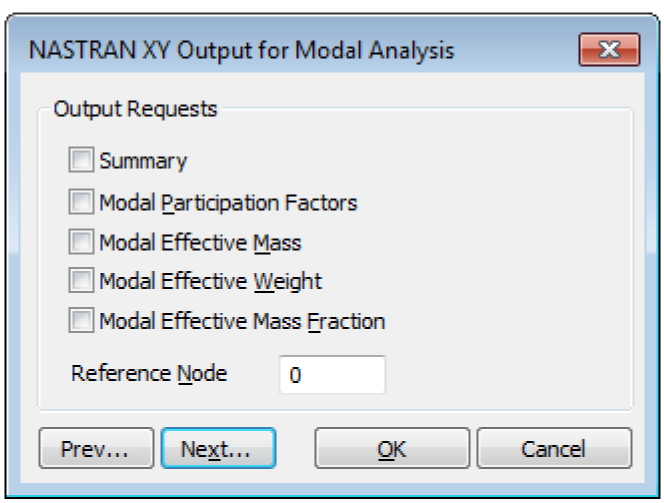
Additional Rotor Dynamics Options

Essentially, each of the other sections of this dialog box specify field(s) on the ROTORD entry for NX Nastran. Choosing an option in the dialog box will write out the appropriate text or number to the corresponding field.

- **Reference System** - enters either “FIX” or “ROT” to the REFSYS field.
- **Rotor Speed** - *Start Value* = RSTART field; *Step Size* = RSTEP field; *Number of Steps* = NUMSTEP field.
- **Speed Input Units** - writes “RPM”, “CPS”, “HZ”, or “RAD” to the RUNIT field.
- **Output Units** - writes “RPM”, “CPS”, “HZ”, or “RAD” to the FUNIT field.
- **Printed Output -** writes “0” (*None*), “1” (*Generalized Matrices*), “2” (*Eigenvalue Summary/Eigenvectors*), or “4” (*Both*) to the ROTPRT field.
- **Mode/Whirl Output -** writes “0.0” (None), “-1.0” (*All RPM*), or “>0.0” (*Rotor Speed* value) to CMOUT field.
- **Other Options -** When “checked”, the *Steiner Inertia* will write YES to the ZSTEIN field (NO if “unchecked”, which is the Default), while *Whirl Threshold* will write a value to the ORBEPS field.
- **Response Calc**, **Excitation**, and **Excitation Order** - used for Model Frequency Response (SOL 111) and represent the SYNC, ETYPE, and EORDER fields on the ROTORD.
- **Modes for Dynamic Response** (SOL 111 only) - writes appropriate MODSEL and SET Case Control lines.

8.7.1.12 NASTRAN XY Output for Modal Analysis

The *NASTRAN XY Output for Modal Analysis* dialog box appears if you pick the *Modal* solution type on the *Nastran Dynamic Analysis* dialog box for the following solution types: Normal Modes/Eigenvalue, Random, and Buckling. It also applies to Transient Dynamic/Time History and Frequency/Harmonic response when the system modes are calculated.



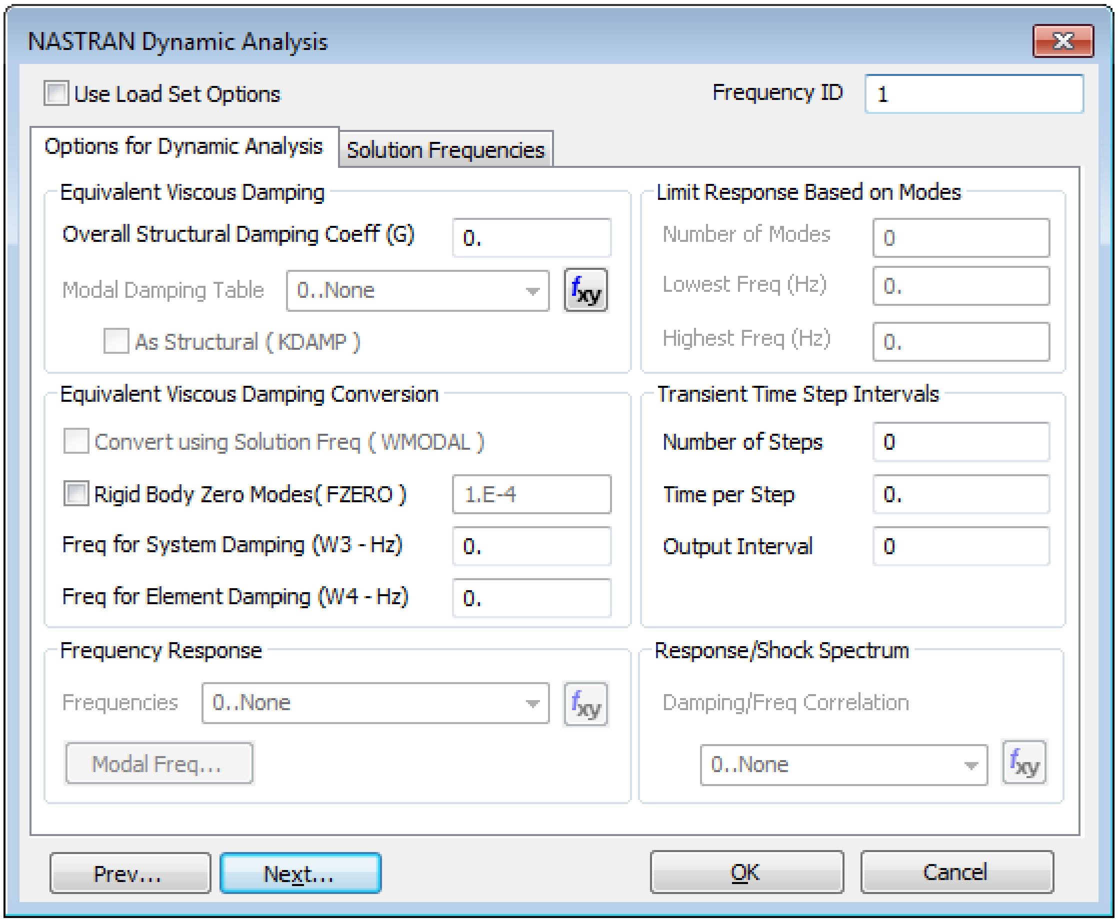
This dialog box controls the type of modal participation information that is written to the PRINT output file (*.f06).

If you enter a *Reference Node,* Nastran will use it for the calculation. If you leave the value as 0, Nastran will use the origin of the global rectangular coordinate system.

FEMAP will read the output information into a FEMAP function. In FEMAP, you can display this data as an XY plot using the *Charting* dockable pane.

8.7.1.13 NASTRAN Direct Transient Analysis

The *NASTRAN Dynamic Analysis* dialog box provides the control information for Direct Transient dynamic analyses. *Direct* or *Modal* is chosen in the *NASTRAN Modal Analysis* dialog box of a the *Analysis Set Manager* when *Analysis Type* is set to “3..Transient Dynamic/Time History” for an analysis set. The *Use Load Set Options* check box can be used to ignore this dialog box and instead use the options set with the *Model, Load, Dynamic Analysis* command. See Section 4.3.5.2, "Model, Load, Dynamic Analysis..."



Equivalent Viscous Damping

This box provides damping information for the structure. Only the *Overall Structural Damping Coefficient* may be input for Direct Transient analysis. *Overall Structural Damping Coefficient* creates the PARAM,G entry

Equivalent Viscous Damping Conversion

Information for both system damping and element damping is provided in this box. These values provide the conversion from the frequency domain, in which damping is usually defined, into the time domain. The *Frequency for System Damping (W3 - Hz)* is divided into the overall damping coefficient, then multiplied by the stiffness to obtain element (or stiffness) damping. The *Frequency for Element Damping (W4 - Hz)* is used in combination with the material damping values to obtain structural damping.

Specify *Rigid Body Zero Modes (FZERO)* to have modes with values under specified value be considered “0”.

The *Rigid Body Zero Modes (FZERO)* creates PARAM,FZERO entry, *Frequency for System Damping (W3 - Hz)* creates the PARAM,W3 entry, while The *Frequency for Element Damping (W4 - Hz)* creates PARAM,W4.

Transient Time Step Intervals

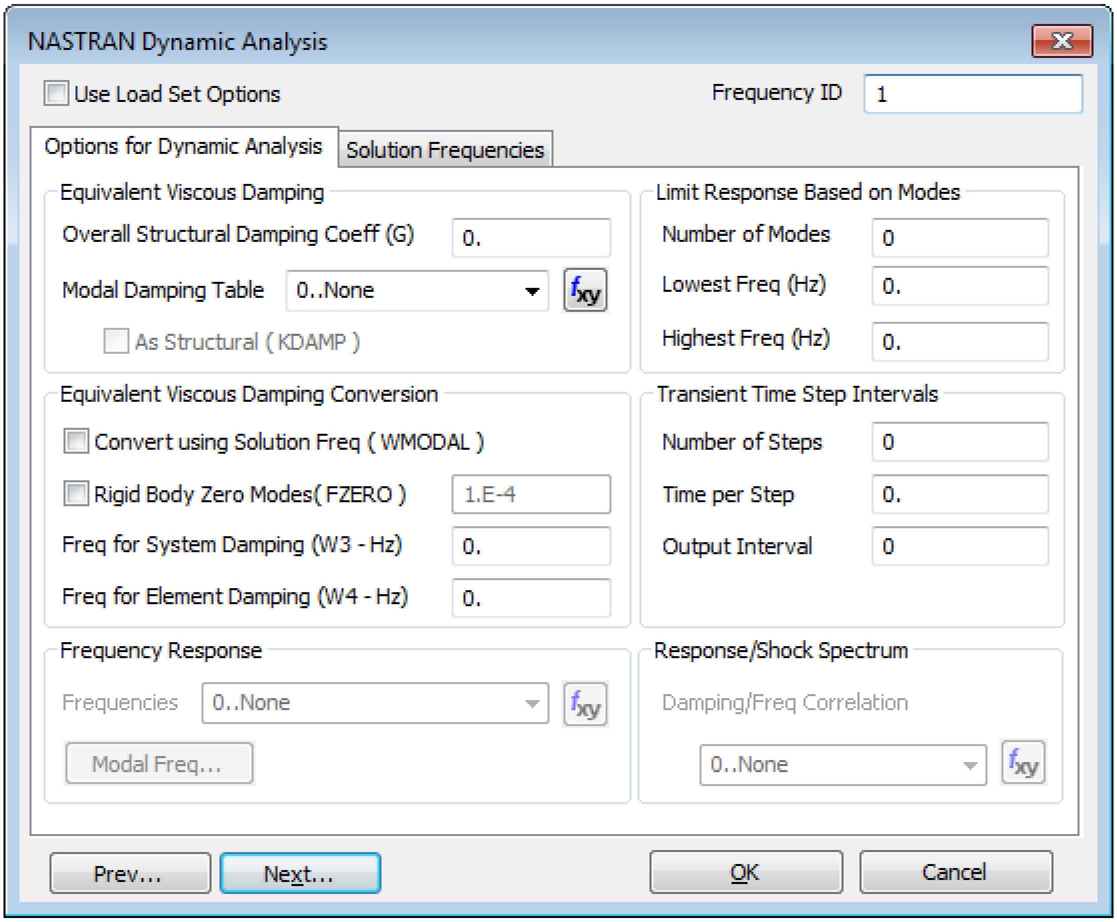
These options control the number of steps, size of steps, and the output interval. Creates the TSTEP entry.

Solution Frequencies tab

The *Solution Frequencies* tab is not used for Direct Transient analysis.

8.7.1.14 NASTRAN Modal Transient Analysis

The *NASTRAN Dynamic Analysis* dialog box provides the control information for Modal Transient dynamic analyses. *Direct* or *Modal* is chosen in the *NASTRAN Modal Analysis* dialog box of a the *Analysis Set Manager* when *Analysis Type* is set to “3..Transient Dynamic/Time History” for an analysis set. The *Use Load Set Options* check box can be used to ignore this dialog box and instead use the options set with the *Model, Load, Dynamic Analysis* command. See Section 4.3.5.2, "Model, Load, Dynamic Analysis..."



Note: When using the *Modal* option, the Frequency Range may be set in the *Range of Interest* section of the *NASTRAN Modal Analysis* dialog box. The number of modes to extract may be specified using *Number Desired* in the *Eigenvalues and Eigenvectors* section. See Section 8.7.1.9, "NASTRAN Modal Analysis" for more information.

Equivalent Viscous Damping

This box provides damping information for the structure. Either the *Overall Structural Damping Coefficient* along with a value for *Freq for System Damping (W3 - Hz)* or a *Modal Damping Table* may be input for Model Transient analysis. *Overall Structural Damping Coefficient* creates the PARAM,G entry, while the *Modal Damping Table* will create the SDAMPING entry in the Case Control section and a TABDMP1 entry in the Bulk Data section.

Equivalent Viscous Damping Conversion

Information for both system damping and element damping is provided in this box. These values provide the conversion from the frequency domain, in which damping is usually defined, into the time domain. When checked, Convert using Solution Freq (WMODAL) specifies a structural-to-viscous damping conversion method that uses the solved modal frequencies as conversion factors. Specify *Rigid Body Zero Modes (FZERO)* to have modes with values under specified value be considered “0”. The *Frequency for System Damping (W3 - Hz)* is divided into the overall damping coefficient, then multiplied by the stiffness to obtain element (or stiffness) damping. The *Frequency for Element Damping (W4 - Hz)* is used in combination with the material damping values to obtain structural damping.

Convert using Solution Freq (WMODAL) creates PARAM,WMODAL,YES entry, *Rigid Body Zero Modes (FZERO)* creates PARAM,FZERO entry, *Frequency for System Damping (W3 - Hz)* creates the PARAM,W3 entry, while The *Frequency for Element Damping (W4 - Hz)* creates PARAM,W4.

Limit Response Based on Modes

These options allow you to limit the modes used to analyze the response of the structure by allowing you to set a subset of the frequency range specified in the *NASTRAN Modal Analysis* dialog box or simply enter a fewer number of modes to use. This can be useful if restarting from a *Modal Analysis* which had a larger frequency range or more modes than are needed to run an accurate *Modal Transient* analysis.

Number of Modes will write the PARAM,LMODES entry, *Lowest Freq (Hz)* will write PARAM,LFREQ and *Highest Freq (Hz)* will write PARAM,HFREQ.

Transient Time Step Intervals

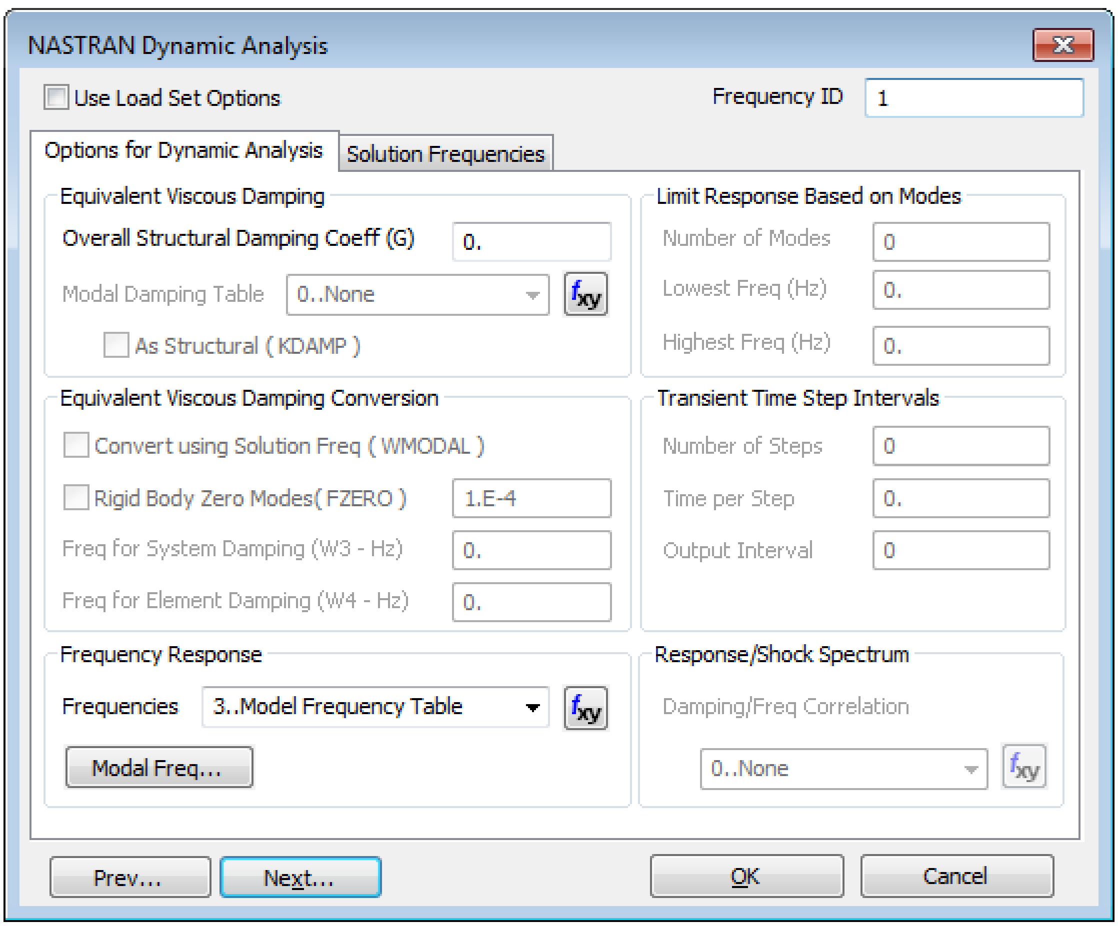
These options control the number of steps, size of steps, and the output interval. Creates the TSTEP entry.

Solution Frequencies tab

The *Solution Frequencies* tab is not used for Modal Transient analysis.

8.7.1.15 NASTRAN Direct Frequency Analysis

The *NASTRAN Dynamic Analysis* dialog box provides the control information for Direct Frequency response analyses. *Direct* or *Modal* is chosen in the *NASTRAN Modal Analysis* dialog box of a the *Analysis Set Manager* when *Analysis Type* is set to “4..Frequency/Harmonic Response” for an analysis set. The *Use Load Set Options* check box can be used to ignore this dialog box and instead use the options set with the *Model, Load, Dynamic Analysis* command. See Section 4.3.5.2, "Model, Load, Dynamic Analysis..."



Frequency ID

This value is used to specify the ID of the Frequency Set which is written to Case Control as FREQUENCY=SID and used as the Set ID (SID) by all lists of solution frequencies (FREQ and FREQi entries) for this Analysis Set.

Equivalent Viscous Damping

This box provides damping information for the structure. Only the *Overall Structural Damping Coefficient* may be input for Direct Frequency analysis. *Overall Structural Damping Coefficient* creates the PARAM,G entry.

Frequency Response

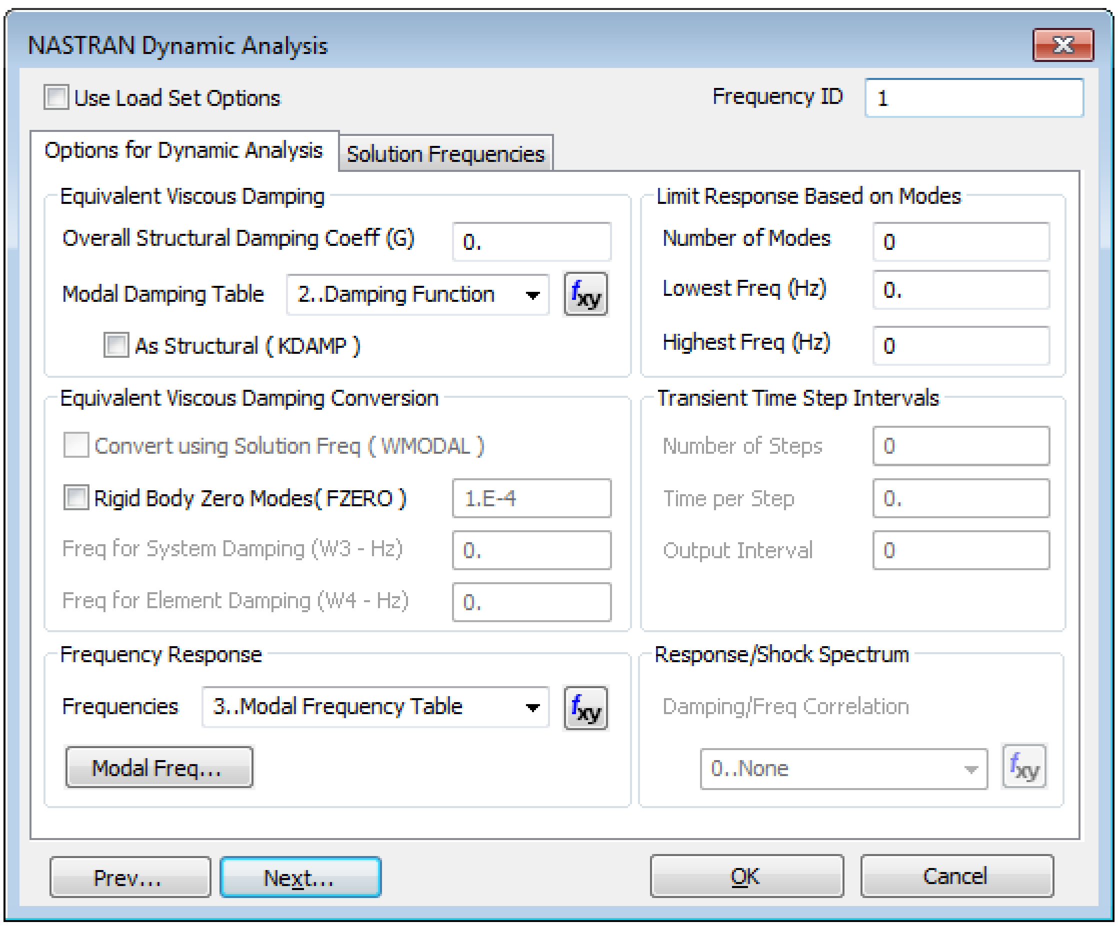
This section allows you to specify the Frequencies where you would like to calculate response. When performing a Direct Frequency analysis, this is simply a “vs. Frequency” function containing all the frequencies in the X column and a Factor (usually 1.0) in the Y column. Creates the FREQUENCY entry in Case Control and the FREQ entry in the Bulk Data section.

Solution Frequencies tab

The *Solution Frequencies* tab in Direct Frequency Analysis may be used to define the *Solution Frequencies* in an alternate way for the analysis. Only “0..FREQ”, “1..FREQ1”, and “2..FREQ2” lists of solution frequencies should be selected for Direct Frequency Analysis. For more information on creating lists of solution frequencies, see "Solution Frequencies tab" in Section 8.7.1.16, "NASTRAN Modal Frequency Analysis"

8.7.1.16 NASTRAN Modal Frequency Analysis

The *Dynamic Control Options* dialog box provides the control information for Modal Frequency dynamic analyses. *Direct* or *Modal* is chosen in the *NASTRAN Modal Analysis* dialog box of a the *Analysis Set Manager* when *Analysis Type* is set to “4..Frequency/Harmonic Response” for an analysis set. The *Use Load Set Options* check box can be used to ignore this dialog box and instead use the options set with the *Model, Load, Dynamic Analysis* command. See Section 4.3.5.2, "Model, Load, Dynamic Analysis..."



Note: When using the *Modal* option, the Frequency Range may be set in the *Range of Interest* section of the *NASTRAN Modal Analysis* dialog box. The number of modes to extract may be specified using *Number Desired* in the *Eigenvalues and Eigenvectors* section. See Section 8.7.1.9, "NASTRAN Modal Analysis" for more information.

Frequency ID

This value is used to specify the ID of the Frequency Set which is written to Case Control as FREQUENCY=SID and used as the Set ID (SID) by all lists of solution frequencies (FREQ and FREQi entries) for this Analysis Set.

Equivalent Viscous Damping and Equivalent Viscous Damping Conversion

This box provides damping information for the structure. Either the *Overall Structural Damping Coefficient* along with a value for *Freq for System Damping (W3 - Hz)* or a *Modal Damping Table* may be input for Model Frequency analysis. *Overall Structural Damping Coefficient* creates the PARAM,G entry, while the *Modal Damping Table* will create the SDAMPING entry in the Case Control section and a TABDMP1 entry in the Bulk Data section. When checked, the *As Structural* check box will write out PARAM,KDAMP,-1.

Specify *Rigid Body Zero Modes (FZERO)* to have modes with values under specified value be considered “0”.

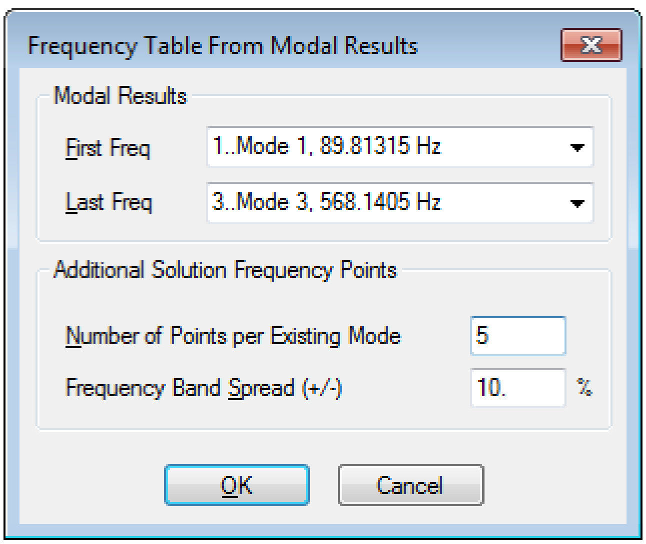
Limit Response Based on Modes

These options allow you to limit the modes used to analyze the response of the structure by allowing you to set a subset of the frequency range specified in the *NASTRAN Modal Analysis* dialog box or simply enter a fewer number of modes to use. This can be useful if restarting from a *Modal Analysis* which had a larger frequency range or more modes than are needed to run an accurate Modal Frequency analysis.

Number of Modes will write the PARAM,LMODES entry, *Lowest Freq (Hz)* will write PARAM,LFREQ and *Highest Freq (Hz)* will write PARAM,HFREQ.

Frequency Response

This section allows you to specify the frequencies where you would like to calculate response. If you have previously performed a modal analysis on your model, and have the solution information in the current model, you can automatically create a solution frequencies table from that output. Simply press *Modal Freq*, and you will see the *Frequency Table From Modal Results* dialog box.



The modal frequency in each output case will be selected for the *Solution Frequency* table. Additionally, frequencies in a band near each modal frequency can be chosen using the *Additional Solution Frequency Points* section. The *Number of Points per Existing Mode* defines the number of frequencies to be included for each modal frequency, while the *Frequency Band Spread* defines the placement of the additional frequencies.

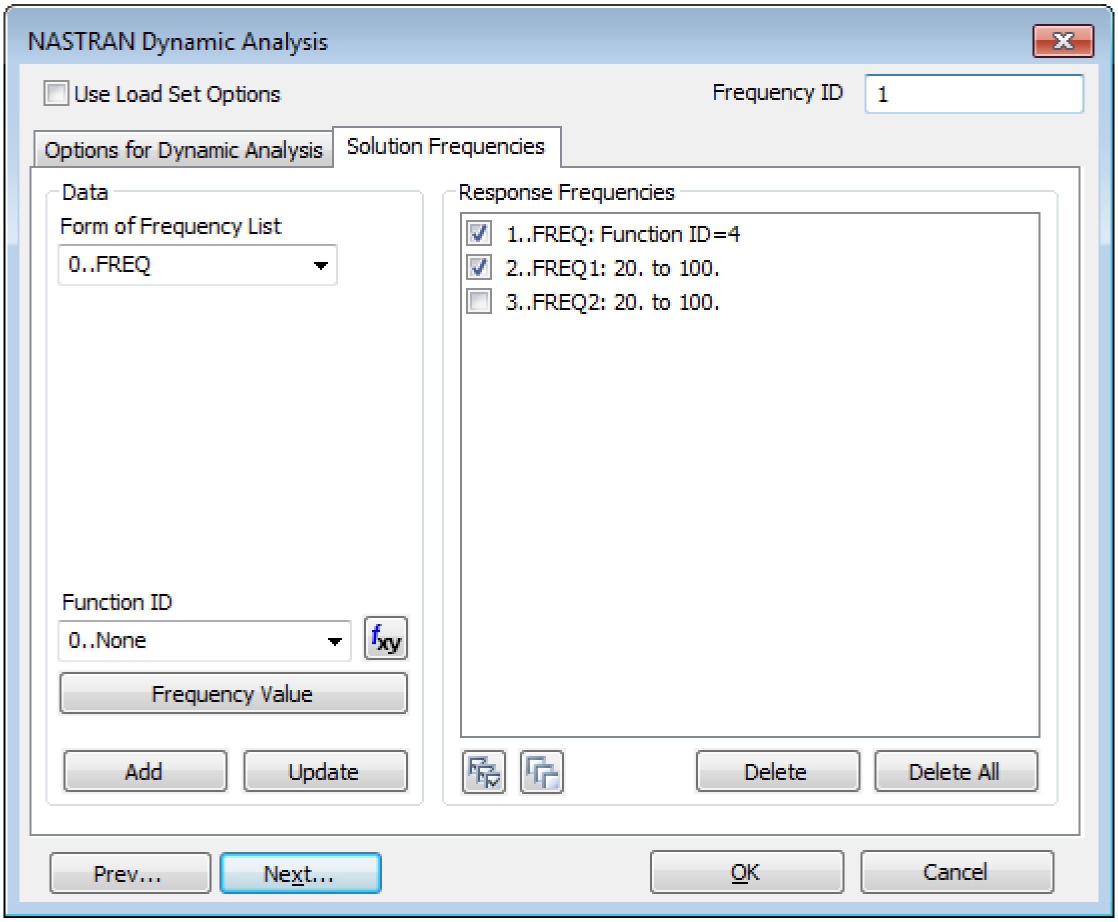
Choosing only one point per mode will select just the modal frequencies. Choosing three points per mode will select the modal frequencies and two additional frequencies at the modal frequency plus and minus the spread value. The number of points must always be odd so that the modal frequencies themselves are selected.

Solution Frequencies tab

The *Solution Frequencies* tab may be used to define the “solution frequencies” in an alternate way for direct frequency response, modal frequency response, and random response analysis.

This tab offers two sections: *Data*, which is used to specify each list of solution frequencies, and *Response Frequencies*, where, once added, each list of solution frequencies will appear and can be chosen for use in an analysis.

Once a list of solution frequencies has been created, you can select it for use in any Analysis Set where the *Solution Frequencies* tab is available. Only items which are "checked" in the *Response Frequencies* list for an Analysis Set will be written to the Nastran input file.



Data

The options available to specify a list of solution frequencies correspond to the various entries Nastran has to define a frequency list (FREQ, FREQ1, FREQ2, FREQ3, FREQ4, and FREQ5). In all cases, you need to click the *Add* button to have the list appear in the *Response Frequencies* list.

Form of Frequency List

This drop-down is used to select the type of "Nastran Frequency List" to create. The inputs needed to define each type of frequency list vary based on the selected option.

The types of frequency list are:

- ***FREQ*** - Can be defined by selecting a function from the *Function ID* drop-down (only X values will be used) or by clicking the *Frequency Value* button, which displays the *Frequency Response Input* dialog box. In either case, the entered values are used as the solution frequencies by Nastran.

  When *Frequency Division* is set to "Custom", enter values directly into the *Frequency Value (Fi)* field. Values MUST be greater than or equal to 0.0. Click the *Add* button to add the current value in *Frequency Value (Fi)* to the list of values. Click *Update* button to change a highlighted value to the value currently in the *Frequency Value (Fi)* field. Click *Delete* button to remove the value from the list. The *Reset* button can be used to clear all values from the list. The *Copy to Clipboard* and *Paste from Clipboard* icon buttons can be used to copy/paste the current list of values to the clipboard or paste values from the clipboard into the dialog box, respectively.

  When *Frequency Division* is set to "Bias", enter a *Number*, choose a type of *Bias* ("Bias Equal", "Bias at Start", "Bias at End", "Bias at Center", or "Bias at Both Ends"), enter a *Bias Factor* (if needed), and a *Range* (lowest value and highest value). Once all parameters have been specified, click the *Add* button to add values.

- ***FREQ1*** - Enter a *First Frequency (F1)*, *Number of Increments (NDF)*, and *Frequency Increment (DF)*.

For example *F1* = 20.0, *NDF* = 5, and *DF* = 20.0 would produce a list with values of 20, 40, 60, 80, 100, and 120.

- ***FREQ2*** - Enter a *First Frequency (F1)*, *Last Frequency (F2)*, and *Number of Log Interval (NF)*.

  For example *F1* = 20, *F2* = 100, and *NF* = 4 would produce a list with values of 20, 29.907, 44.7214, 66.874, and 100.

- ***FREQ3*** - Enter *First Frequency (F1)*, *Last Frequency (F2)*, *Number of Frequencies (NEF)* (number of excitation frequencies between two modal frequencies, value includes both modal frequencies), and *Cluster* (factor used for "clustering" of excitation frequencies near the end points of range). Also, you can choose to use *Logarithmic* (checked) or *Linear* (unchecked) interpolation between frequencies.

  For example, *F1* = 20, *F2* = 300, *NEF* = 4, and *Cluster* = 1.0, *Logarithmic* not checked, and 2 Modes with frequency values 89.8135 and 243.5258 would solve using a list of **20**, 43.271, 66.5421, **89.8131**, 141.051, 192.288, **243.526**, 262.351, 281.175, and **300**. **Bold** values are first, last, and 2 modal values.

- ***FREQ4*** - Enter *First Frequency (F1)*, *Last Frequency (F2)*, *Number of Frequency (NFM)* (number of evenly spaced frequencies per "spread" mode), and *Spread (FSPD, +/-%)* (specified as a % and is the "frequency spread", +/- the fractional amount, for each mode).

  For example, *F1* = 20, *F2* = 300, *NFM* = 5, and *FSPD, +/-%* = 3, with 2 Modes of 89.8135 (Mode 1) and 243.5258 (Mode 2) would produce a list of 87.1188 (97% of Mode 1 value), 88.4659 (98.5%), 89.8135 (100%), 91.1603 (101.5%), 92.5075 (103%), 236.22 (97% of Mode 2 value), 239.873 (98.5%), 243.5258 (100%), 247.179 (101.5%), and 250.832 (103%).

- ***FREQ5*** - Enter *First Frequency (F1)* and *Last Frequency (F2)*. Only modes which fall within the frequency range between *F1* and *F2* will be used. Now enter "fractions" of each mode to use by selecting a function from the *Function ID* drop-down (only X values will be used) or by clicking the *Frequency Fraction* button, which displays the *Frequency Response Input* dialog box. This dialog box is similar to the one used when creating a list for *FREQ*, only you enter values as *Frequency Fraction (FRi)* values instead of actual frequency values. Entering a value of 1.0 will create a value equal to 100% of each modal value in the specified range, 0.9 will create a value 90% of each modal value, while 1.05 will create a value 105% of each modal value.

  For example, *Frequency 1* = 20, *Frequency 2* = 300, with 2 Modes of 89.8135 (Mode 1) and 243.5258 (Mode 2). Using values of 0.9, 1.0, and 1.05 for the *Frequency Fraction (FRi)* would produce a list with 80.8318 (90% of Mode 1 value), 89.8135 (100%), 94.3038 (105%), 219.173 (90% of Mode 2 value), 243.5258 (100%), and 255.702 (105%).

Add Button

Click this button to add a new list of solution frequencies to the *Response Frequencies* list.

Update Button

When an item is highlighted in the *Response Frequencies* list, it will appear in the *Data* section. You can now make changes, then click this button to update a current list of solution frequencies.

Response Frequencies

This list shows all available lists of solution frequencies currently in the model. To select a list of solution frequencies, click the check box to the left of the ID. Only "checked" items in a particular Analysis Set will be written to the Nastran input file and the "checked" items can vary from Analysis Set to Analysis Set. To "check" all available lists of solution frequencies, click the *Select All* icon button or click the *Select None* icon button to "uncheck" all.

Only one item at a time can be highlighted in the *Response Frequencies* list. When highlighted, the data for that list of solution frequencies will appear in the *Data* section and can be updated. Click the *Delete* button to delete the highlighted item or click *Delete All* to delete all lists of solution frequencies.

8.7.1.17 NASTRAN Response Spectrum Analysis

Response spectrum analysis is an approximate method for predicting the peak responses of a transient excitation applied to a simple structure or component.

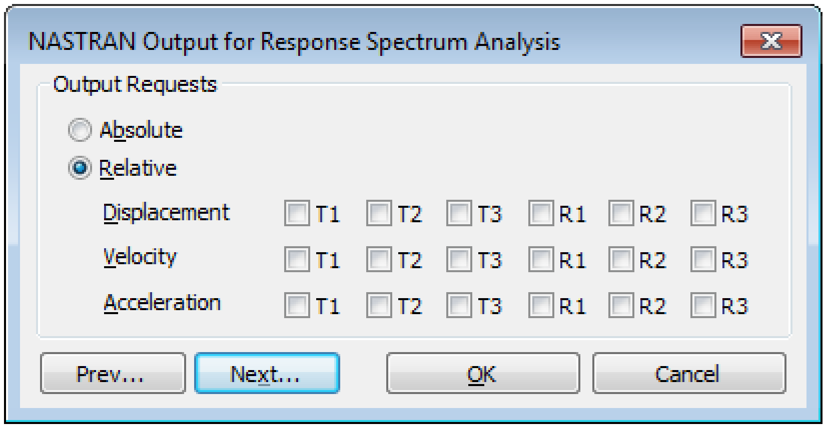
There are two basic steps to determining peak responses:

1. Run the Response Spectrum analysis to create spectrum. You then use the spectrum as input to a Normal Modes analysis.

2. Run a Normal Modes analysis.

Before you run the response spectrum analysis, create a group of nodes for the analysis.

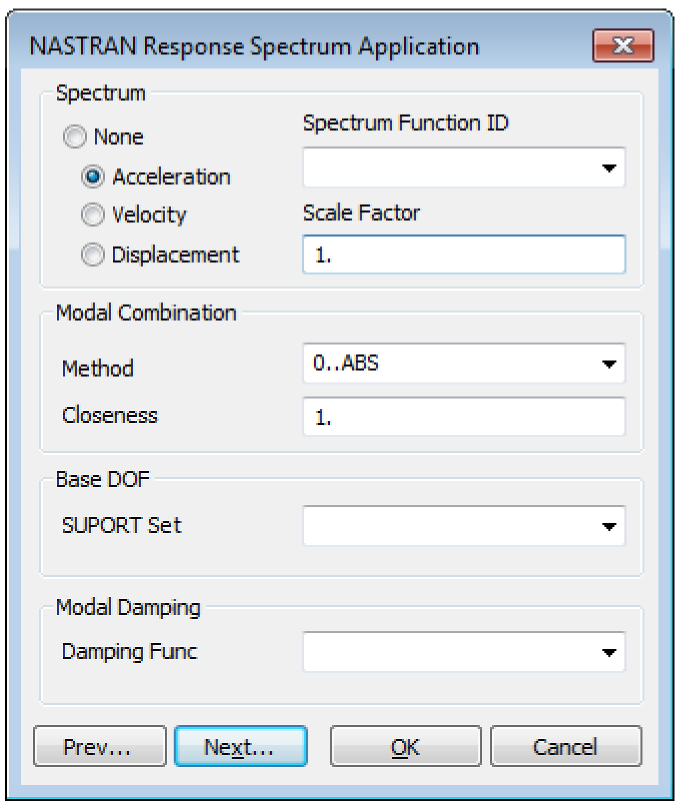
To set up the Response Spectrum analysis, use the *NASTRAN Output for Response Spectrum Analysis* dialog to pick the DOF for displacement, velocity, and/or acceleration.



The software will then display the *Nodal Results* dialog box, where you will select the group of nodes to analyze.

NASTRAN Normal Modes/Response Spectrum Application

In a Normal Modes analysis, you can use the *NASTRAN Response Spectrum Application* dialog box to set up a response spectrum analysis. This type of analysis predicts the peak responses of a transient excitation applied to a simple structure or component.



Before you can run the Normal Modes/Response Spectrum analysis:

- Run a Response Spectrum analysis (set the analysis type to *Response Spectrum*) to create the spectrum that you'll need for the Normal Modes/Response Spectrum analysis.
- Use the FEMAP *Model, Function* command to create a function that applies damping to each input spectra.

Once these two steps are complete, you can set up an Normal Modes/Response Spectrum analysis.

Spectrum

Pick acceleration, velocity, or displacement. Use the FEMAP damping/spectra function as the *Spectrum Function*, and enter a scale factor.

Modal Combination

- Pick the method used to combine the peak responses into the overall response:

  ABS: absolute values

  SRSS: square root sum of the square

  NRL: U.S. Navy shock design modal summation

  NRLO: U.S. Navy shock design modal summation (old method)

- *Closeness*: The *Closeness* value applies to all methods except ABS. The natural frequencies that are greater than this value will be calculated using the method that you selected. However, the software will use the ABS method to calculate any natural frequencies that are less than the Closeness value.

Base DOF

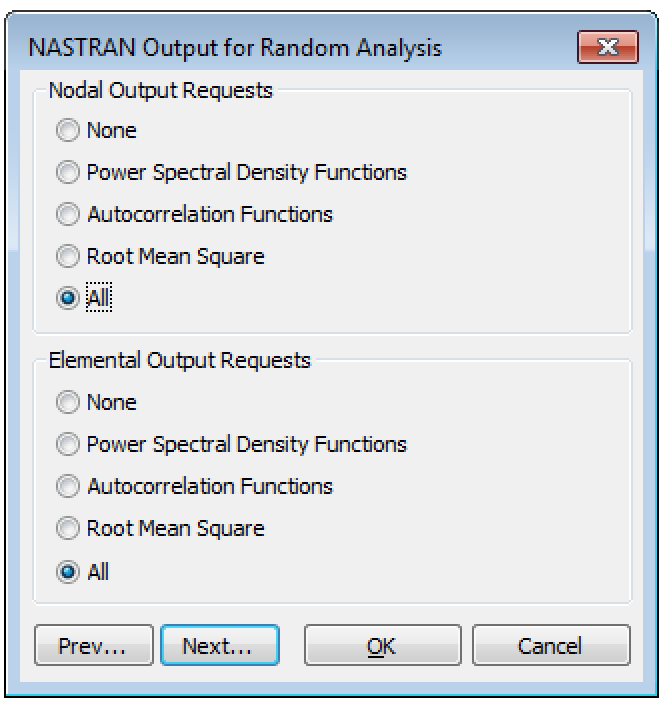
Pick a constraint (SUPORT) set to define the rigid body DOF to be excluded from the analysis.

Modal Damping

To define modal damping for the analysis, you must first create it with the FEMAP *Model, Function* command. To create the function, define the damping value as a function of natural frequency. You can create one of three function types: structural damping vs. frequency, critical damping vs. frequency, or Q damping vs. frequency.

8.7.1.18 NASTRAN Random Response Analysis

For Random Response analyses, you should specify additional output requests. Random Response results are output through the XY plotting routines, as well as separate output requests which are similar to requests for stress, strain, etc. In the first *NASTRAN Output for Random Analysis* dialog box, you can select *Power Spectral Density Functions* (PSD), *Autocorrelation Functions*, *Root Mean Square* results, or *All* for both nodal and elemental output.

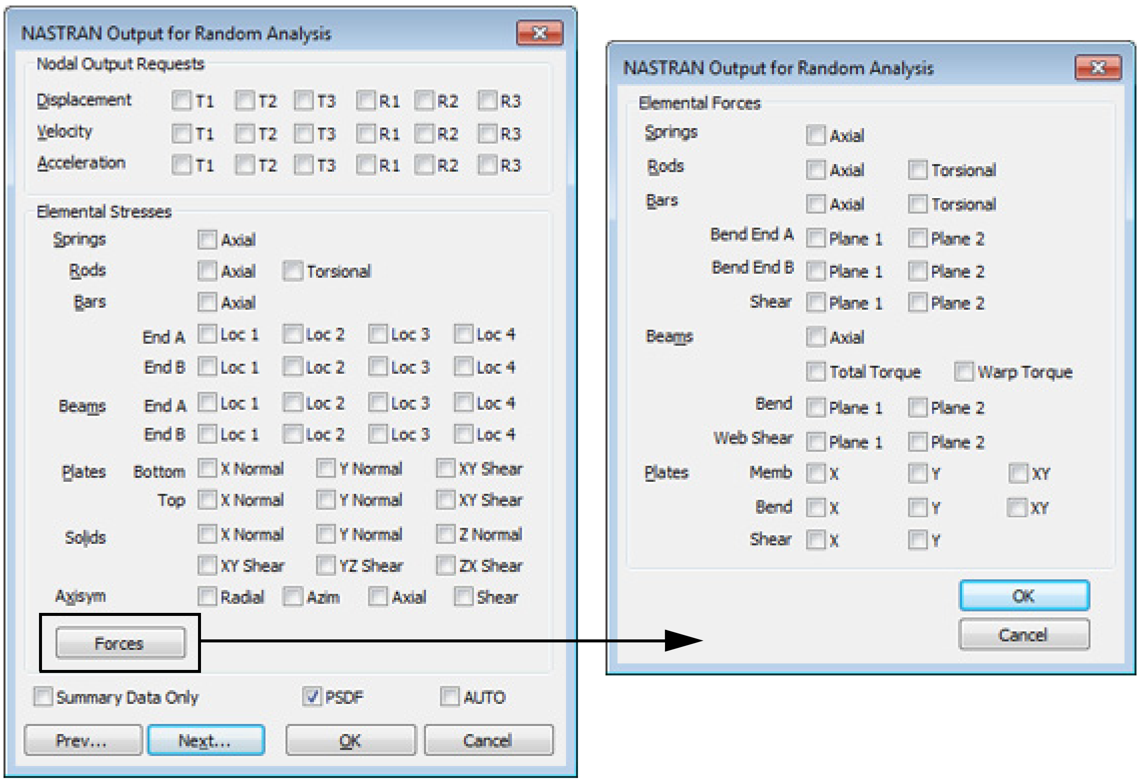


The information you request will then be exported to OUTPUT2 results file (op2).

Note: Imported nodal results from a random analysis (*PSDF*, *RMS*, and *Positive Crossings* output sets) are read “as is” from Nastran Results files. The results are not transformed into the global coordinate system like nodal results from other types of analysis. Therefore, any command which “transforms” nodal output will produce invalid output values for the nodal output vectors in these sets.

Note: Von Mises Stress results from random response analysis are available from NX Nastran version 9.0 and above. They are only found in the Root Mean Square (RMS) output set, so you will need to request *Root Mean Square* or *All* in order to post-process this output vector.

In the second *NASTRAN Output for Random Analysis* dialog box, you can define each individual output for XY plots of random analysis results. You should carefully define which output types are required for your analysis because large amounts of output will be obtained if all output types are requested for a significant portion of a large model.



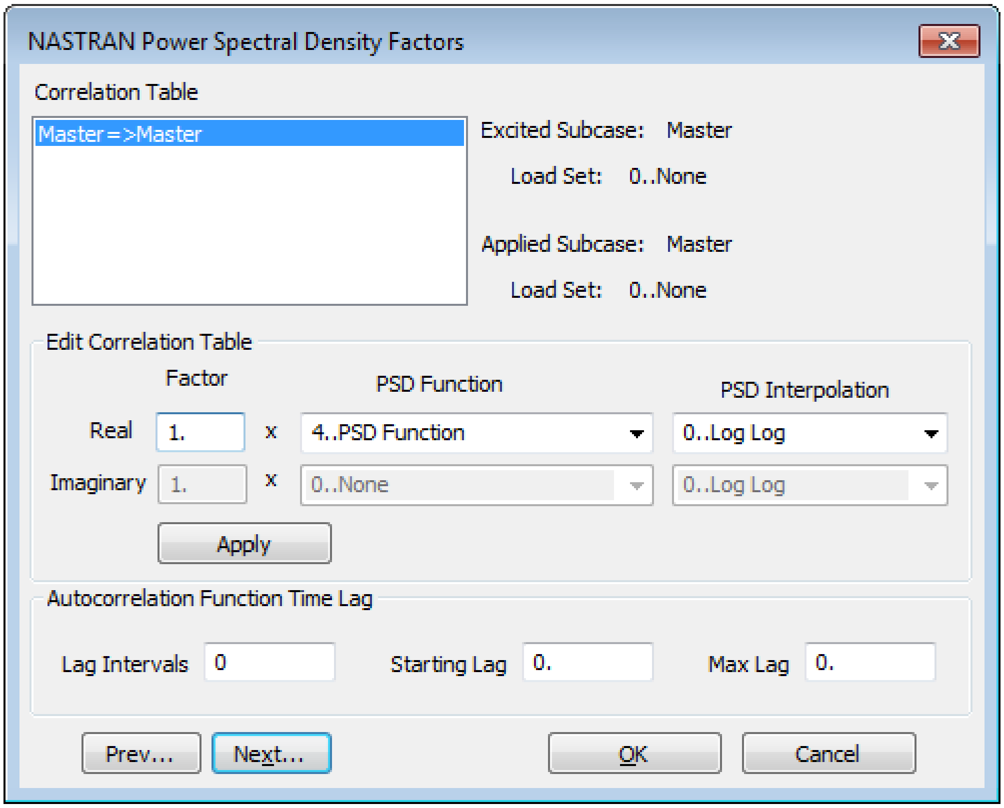
After selecting the type of output, you will then select the nodes and/or elements by selecting an existing group of nodes for items selected in the *Nodal Output Requests* section and/or an existing group of elements for items selected in the *Elemental Stresses* and *Elemental Forces* sections.

You can also choose to recover Power Spectral Density Functions, which is the default, and/or Autocorrelation Functions by checking the box *PSDF* and/or *AUTO*, respectively.

You may also limit the output to *Summary Data Only*. The results will not include individual values vs. frequency. The *Summary Data Only* option is most useful when you need to recover RMS values for your entire model, but do not need the contributions from the individual frequencies.

NASTRAN Power Spectral Density Factors

In a random response analysis, use the *NASTRAN Power Spectral Density Factors* dialog box to define complex cross spectral correlations between the load sets defined for a random response analysis.



The numbers of the *Correlation Table* correspond to the cases that define the loads for your analysis. Where the cases listed are the same (i.e, Master=>Master or 1=>1), pick the case from the table, then enter a real factor and select a PSD function to apply to the case. (You create PSD functions using the FEMAP *Model, Function* command.)

Once *Apply* is clicked to update the *Correlation Table* when the cases are the same (i.e., Master=>Master,1=>1), the information will appear as follows:

“Real” Factor*(“Real” Function ID) + :PSD Interpolation option (“Real”)

When the cases are different (such as 1=>2), you must decide whether you want to correlate the cases. If you don’t enter any values for real and imaginary values, the cases will not be correlated. If you do want to correlate the cases, enter the real and imaginary factors and PSD functions.

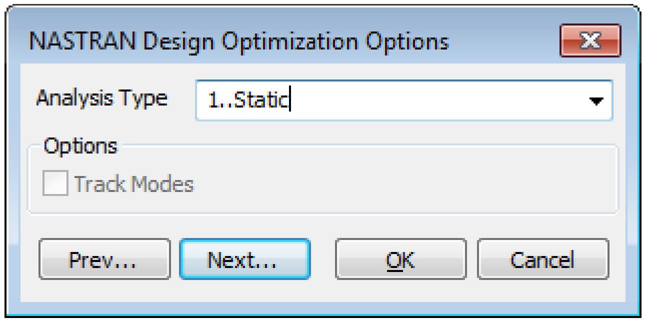
Once *Apply* is clicked to update the *Correlation Table* when the cases are not the same (i.e.,1=>2), the information will appear as follows:

“Real” Factor*(“Real” Function ID) + “Imaginary” Factor*(“Imaginary” Function ID) :PSD Interpolation option (“Real”) :PSD Interpolation option (“Imaginary”)

The items in the *Autocorrelation Function Time Lag* section are used to define time lag constants for use in calculation of autocorrelation functions in random response analysis. All three of these values will be written to the same RANDT1 entry in Nastran, with the value entered for *Lag Intervals* written to the N field, the value for *Starting Lag* written to the T0 field, and the value for Max Lag written to the TMAX field.

8.7.1.19 NASTRAN Design Optimization Options

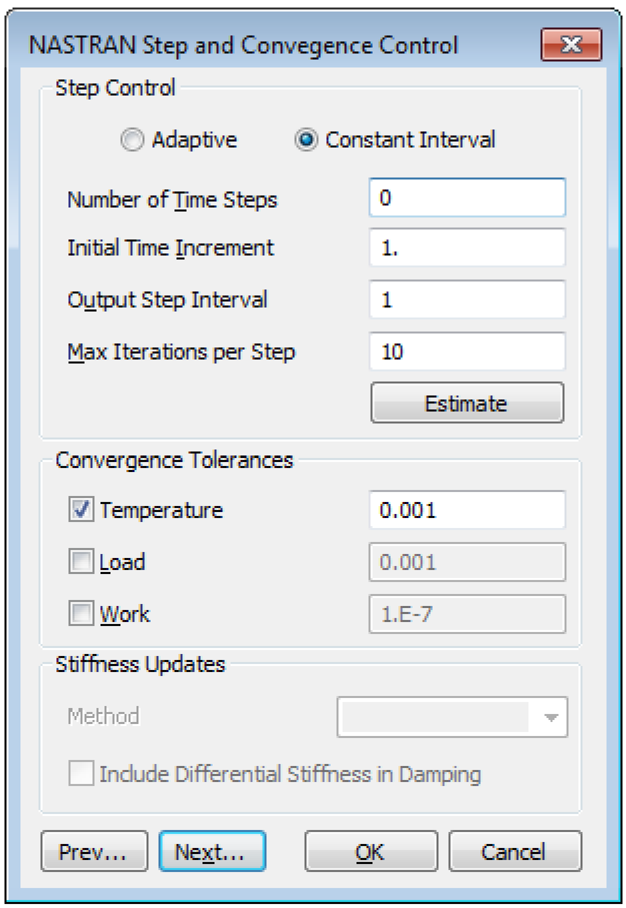
For design optimization analysis, specify the *Analysis Type* as “1..Static” or “2..Normal Modes/Eigenvalue”.



When “2..Normal Modes/Eigenvalue” is selected, an option to Track Modes becomes available. When this option is on, the MODTRACK Case Control entry and corresponding MODTRACK Bulk Data entry will be written to the Nastran input file with a value or 0 for both LOWRNG and HIGHRNG, thereby using all computed modes to search for the designed modes.

8.7.1.20 Heat Transfer Nonlinear Control Options

For heat transfer analysis, you must specify additional iteration control and convergence information.

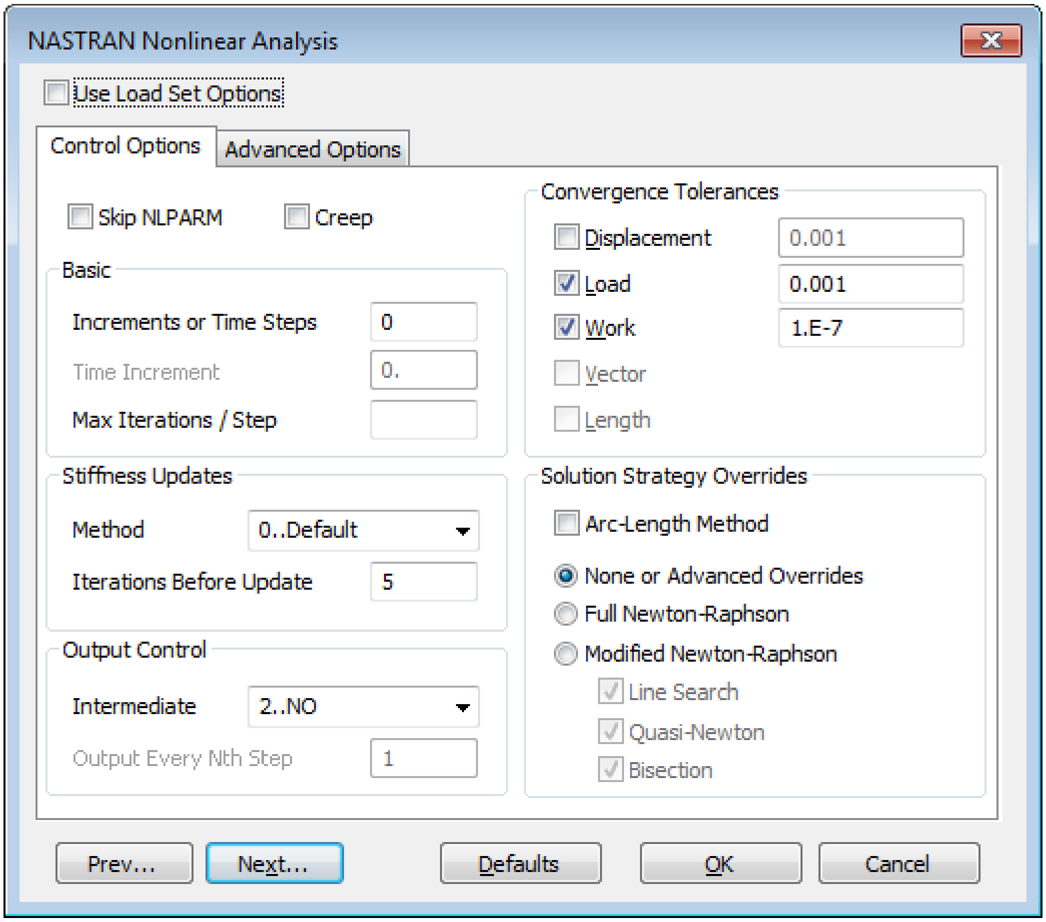


For transient heat transfer, you must input the desired number of time steps, initial time increments, frequency of output and the maximum number of iterations. Your choice of the time stepping and time increment are crucial to proper convergence to an accurate solution. To assist you in defining these values, the *Estimate* button can be used to examine the model that you have defined, including material properties, the duration of any functions, and your mesh, and make a guess at the values for the other options. Remember, this is just an educated guess based on your model. It may not be what you intended to analyze. It is ultimately up to you to set these values appropriately.

You can choose to require convergence on temperature, load, or work criteria within specified values. You can also choose a method for stiffness updates as well as choose to include differential stiffness in damping. For steady state heat transfer, you also have the option of choosing the convergence tolerances, but not the step control.

8.7.1.21 Nonlinear Static, Nonlinear Transient, and Creep

The nonlinear analysis options for Nonlinear Static, Nonlinear Transient, and Creep may be set in the Master case or for each individual subcase. A subset of options in this dialog box are also available when contact is used in MSC Nastran. In either case, the Nonlinear options are controlled by the *Nonlinear Control Options* dialog box:



Skip NLPARM (Enable NLPARM for MSC Nastran Static Analysis)

When this option is checked, all parameters set in this dialog box for a given subcase will be ignored. Instead, the parameters set in the *Nonlinear Control Options* dialog box of the *Master Requests and Conditions* section of the *Analysis Set Manager* will be used. Also, if this dialog box is never accessed for a particular subcase, the options set in *Nonlinear Control Options* of the *Master Requests and Conditions* will be used. This dialog box is also available in the *Master Requests and Conditions* section and individual subcases for static analysis when *Analysis Program* is set to MSC Nastran, but should only be used if the model contains linear contact. In that case, this option is called *Enable NLPARM* and must be turned on for the subset of available options to be written to the input file.

Creep option (Nonlinear Static analysis only)

Used to indicate if a subcase is a “Creep” subcase (checked) or a “Nonlinear Static” subcase (unchecked).

To perform a creep analysis, two analysis subcases must exist, which reference the same load set (or two load sets with identical load values). The first subcase should be a “standard” Nonlinear Static subcase (i.e., *Creep* unchecked). The second subcase must have *Creep* checked and a value entered for *Time Increment* in the *Basic* section. Also, values should be defined on the *Creep* tab for the material(s) in the model to create accurate results.

Basic

These values provide the time and iteration control information for the nonlinear analysis steps. They control the *Number of Increments* and the *Time Increment* to be used, as well as the *Maximum Iterations* for each step. No time increment is used for static analysis.

Stiffness Updates

This specifies the number of iterations to be performed before the stiffness matrix is updated, as well as the update *Method*. Five different update methods are available, but not all are appropriate for all each solution type. If an inappropriate method is selected, the translator will provide an error message and choose the default method.

Output Control

Output Control information allows you to request or eliminate output at intermediate steps (static and creep) or request *Output Every Nth Step* (Nonlinear Transient only).

Convergence Tolerances

The type of *Convergence Tolerances* (*Load, Displacement,* and/or *Work*) as well as the tolerance values themselves are defined in these boxes. MSC Nastran has two additional tolerances, *Vector* and *Length*, which can be enabled.

Solution Strategy Overrides

This area provides you with the capability to further control the strategy that will be employed to converge toward a solution.

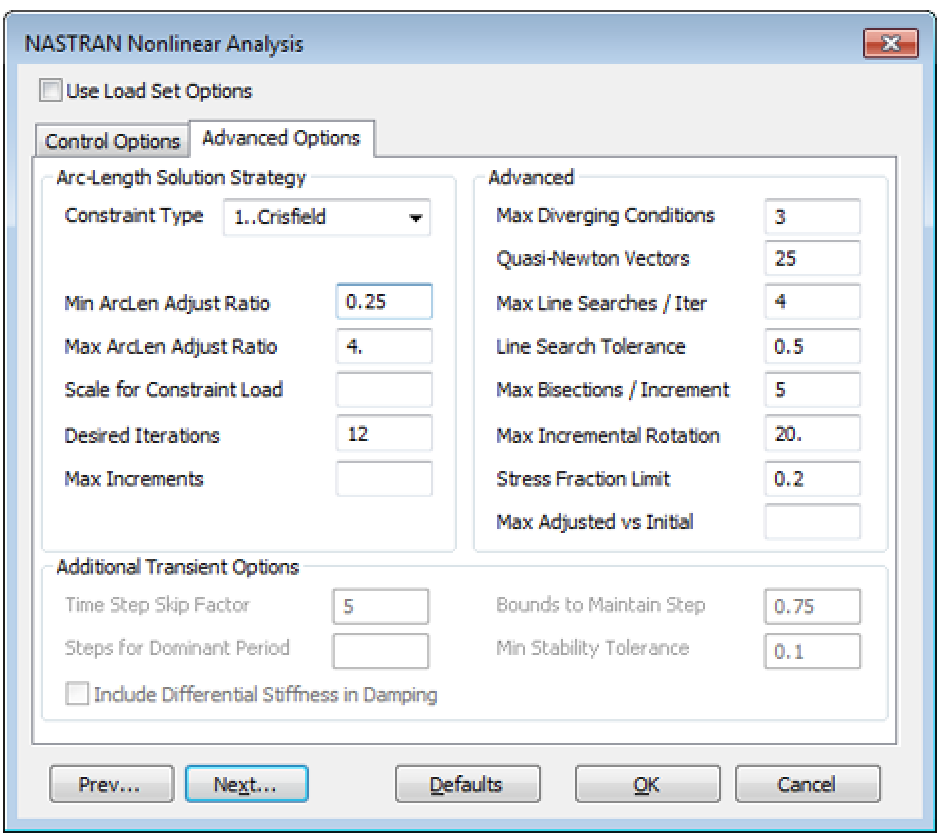
Defaults button

Defaults are automatically set when the dialog box is entered for the first time. Click *Defaults* to reset those values.

Advanced Options tab

This tab enables you to access additional nonlinear analysis options as well as damping inputs for nonlinear transient analysis. For most problems, the nonlinear options are not required, but they are available for experienced analysts to modify the default solution controls.

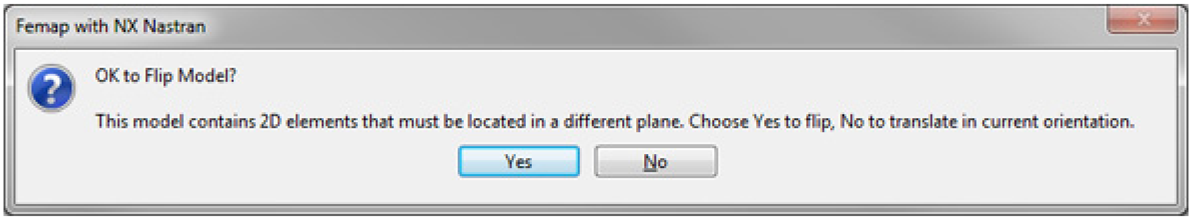
Options in the *Advanced Options* tab, when written to Nastran, are used to define the parameters on the NLPARM statement. In general, FEMAP does not distinguish between blanks and zeros when you enter values into dialog boxes, therefore, when the values are written to Nastran it is normally not possible to control whether a blank or a zero will be written. For some of these fields however this distinction is important, therefore several special cases have been implemented. If you specify a blank, or zero, in the dialog box for any of these cases, you will get a blank in your Nastran file. If you specify a negative value for *Quasi-Newton Vectors*, or for *Max Line Searches/Iter*, you will get a "0". Similarly, if you specify a value that is less than -10, for *Max Bisections / Increment*, you will get a zero. Values less than -10 were chosen because values down to this value are valid for that field.



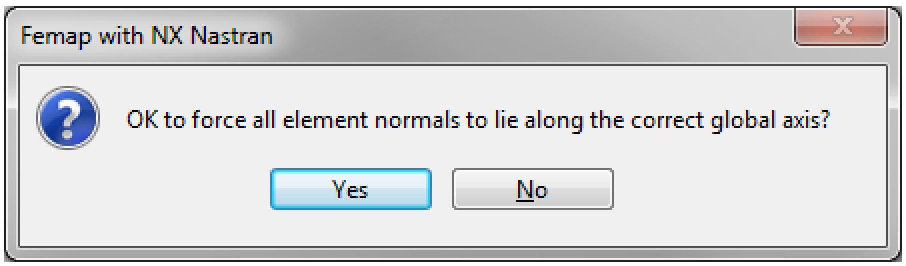
8.7.1.22 Special Notes for Models with Axisymmetric Elements

Since Nastran has some very specific requirements for axisymmetric analyses, FEMAP must ask some additional questions as you translate your model.

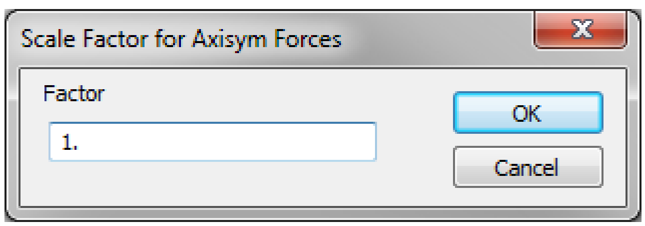
All axisymmetric elements must be located in the global XZ plane (unless you are using hyperelastic elements - then the model must be in the XY plane). FEMAP allows you to build your model in any global plane, and will automatically flip your model to the XZ (or XY for hyperelastic) plane when you answer *Yes* to the above question. You should never answer *No*, unless you already know your model is in the proper plane.



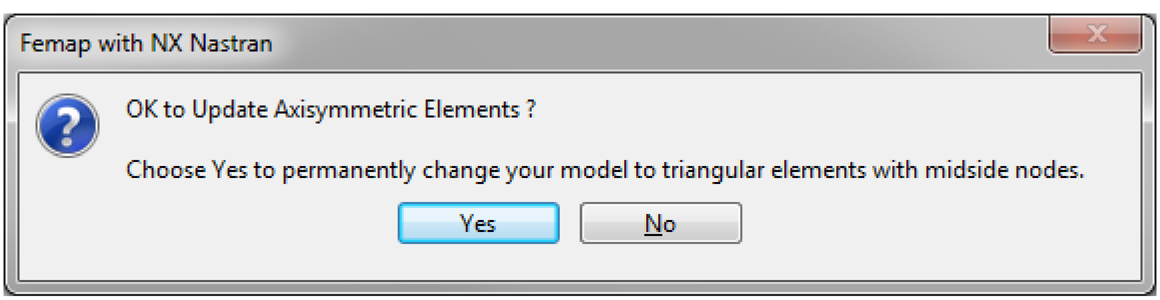
Similarly, all axisymmetric element normals must be consistent. Press *Yes* to automatically check and update your model to the proper conventions.



Some analysis programs require axisymmetric loads be applied as total loads (i.e. the total force on the full circumference), while others apply loads on a per radian basis, and still others on a per unit length basis. Depending on how you have defined your loads, this scale factor lets you translate them with the proper values for Nastran.



Finally, because the preference in Nastran axisymmetric analyses is to use CTRIAX6 elements with midside nodes, FEMAP allows you to automatically convert any elements with no midside nodes to elements with midside nodes by answering *Yes* to this question. This also splits any quadrilateral elements into triangles since there are no quad axisymmetric elements in Nastran. You must therefore press *Yes* if you have quadrilateral elements. If you answer *No*, your elements are not updated. All triangles will still be translated as CTRIAX6 elements with missing midside nodes.



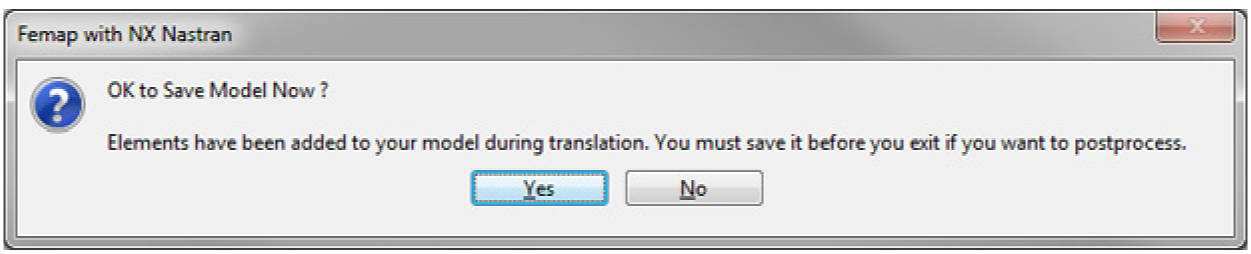
Note: While this automatic update changes the element connectivity to add midside nodes, it cannot automatically update any loads or boundary conditions. This can result in elements that have corners restrained or loaded, but which have midside nodes that are unrestrained and unloaded. Depending on the conditions you are trying to model, this may be incorrect. You should carefully check the modified model and decide whether any changes need to be made prior to completing your analysis.

If you have made any of these changes to your model, you will also be asked to save it after the translation has been completed. To insure compatibility with the results of your analysis for post-processing, you should always save the updated model.

8.7.1.23 Special Notes for Heat Transfer Analysis

If you are performing a heat transfer analysis, the translator not only produces the Nastran file, it also updates your FEMAP model to prepare it for post-processing. Since Nastran uses surface elements (CHBDYi) to apply many thermal loads rather than applying them directly to the structural/conduction elements, most heat transfer analyses require some model modifications. To simplify load definition, FEMAP always allows you to define heat transfer loads directly on the faces of the structural/conduction elements. During the translation however, new surface elements are automatically created in your model as plot-only plate elements. These represent the equivalent CHBYDi elements that were written to the Nastran model.

If you want to successfully post-process the results of your analysis, you **must** save this updated model, since many of the thermal results are reported on the CHBYDi elements. If you do not save these new elements, FEMAP will not be able to display those results.



8.7.1.24 Advanced Nonlinear Analysis (NX Nastran Only)

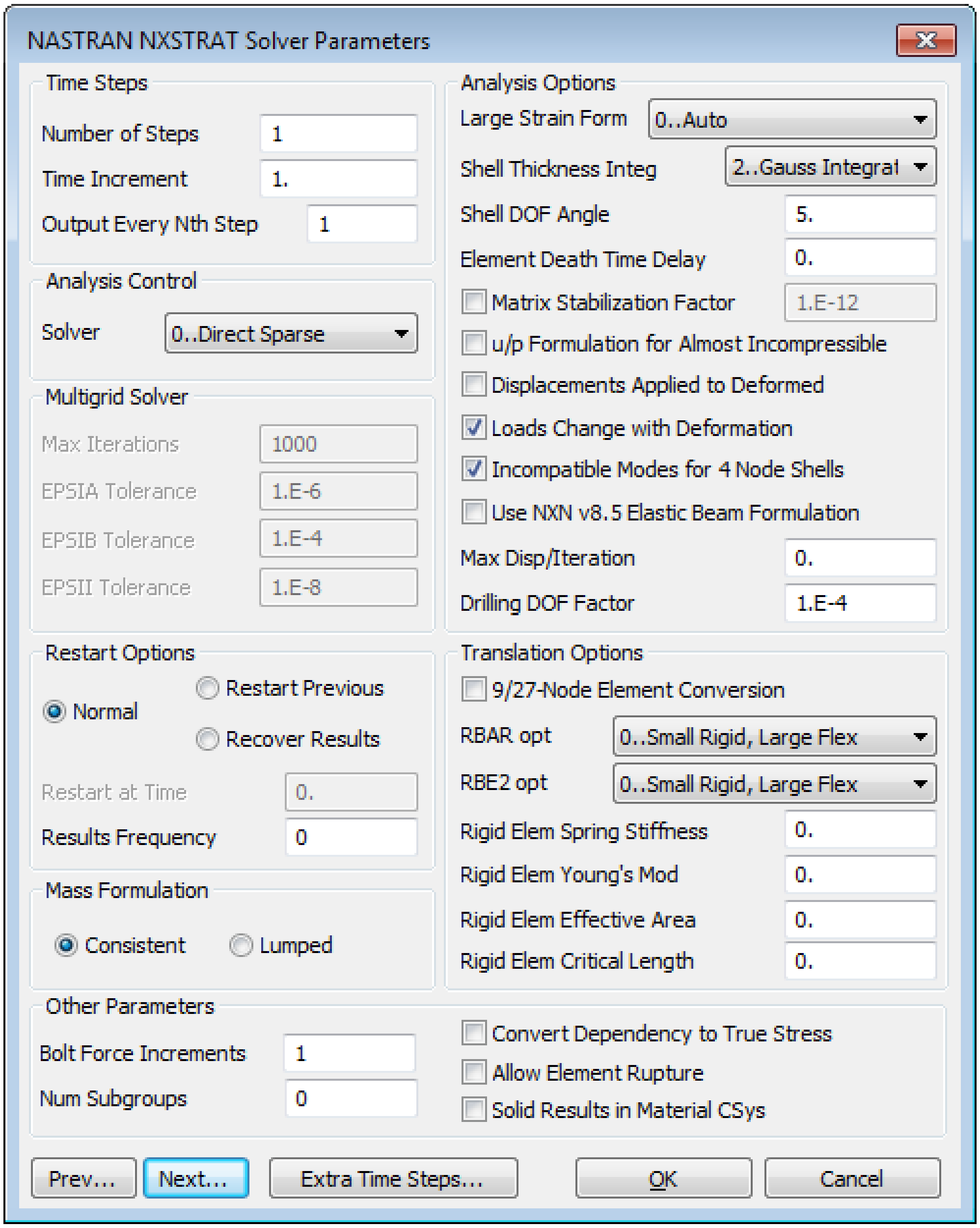
NX Nastran has the ability to perform Advanced Nonlinear Analysis using Solution Sequence 601 (SOL 601).

Specific Solution 601 dialog boxes will appear in the Analysis Set Manager when the Analysis Type has been set to either *22..Advanced Nonlinear Static* or *23..Advanced Nonlinear Transient*. These Dialog boxes contain Overall Solver Parameters (including Extra Time Step definition) and Iteration and Convergence Parameters.

Solution 601 has several “solution specific” materials including various Hyperelastic materials (MATHE - Mooney-Rivlin, Hyperfoam, Ogden, and Arruda-Boyce) and a Gasket material (MATG). These materials are defined using the “Other Types” material type in FEMAP (For more information See Section 4.2.3.6, "Other Types..." in the FEMAP commands manual).

NASTRAN NXSTRAT Solver Parameters

This dialog box allows you to set up Strategy Parameters for SOL 601 (NX Nastran Only).



Time Steps

Number of Steps - number of time steps (of Time Increment value) in transient or nonlinear static analysis using SOL 601. Value is entered into the N1 field on the TSTEP entry.

Time Increment - Time increment for steps in transient or nonlinear static analysis using SOL 601. Value is entered into DT1 field on the TSTEP entry.

Output Every Nth Step - Skip factor for output. Creates NO1 field on TSTEP entry. Every NOi-th step will be saved for output.

Extra Time Steps button: Allows you to enter additional time step options for certain types of analysis. These extra time steps are written out on the TSTEP entry as additional Ni, DTi, and NOi fields corresponding to the number of additional time steps.

Analysis Control

Solver - Lets you choose between 0..Direct Sparse (Direct Sparse Solver), 1..Multigrid (Multigrid Solver), or 2..3-D Iterative (3-D Iterative Solver). Creates the SOLVER field on the NXSTRAT entry. The Direct Sparse is the Default Solver.

Multigrid Solver (Only used when SOLVER = 1)

Max Iterations - Maximum number of iterations allowed for the Multigrid solver to converge. Creates the ITE-MAX field on the NXSTRAT entry. Value must be an integer greater than 0, default = 1000.

EPSIA Tolerance - Convergence tolerance EPSIA. Creates EPSIA field on NXSTRAT entry (Default = 1.0E-6)

EPSIB Tolerance - Convergence tolerance EPSIB. Creates EPSIB field on NXSTRAT entry (Default = 1.0E-4)

EPSII Tolerance - Convergence tolerance EPSII. Creates EPSII field on NXSTRAT entry (Default = 1.0E-8)

Restart Options

Restart Previous Analysis - Indicates mode of Execution. When set to *Normal*, which is the default, places a 0 (Normal analysis run, no restarts) in the MODEX field of the NXSTRAT entry. When set to *Restart Previous*, places a 1 (Restart Analysis) in the MODEX field of the NXSTRAT entry. When set to *Recover Results*, places a 2 in the MODEX field of the NXSTRAT entry, which simply recovers the results file without running additional time steps (GPFROCE and SPCFORCES cannot be recovered).

Restart at Time - Solution starting time. Creates the TSTART field on the NXSTRAT entry. IF MODEX=1, TSTART must equal a solution time in which data was saved in the previous run. IF TSTART = 0.0, the last time step in the restart will be used.

Results Frequency - Frequency of saving the analysis results in the restart file. Creates IRINT field on the NXSTRAT entry. Default = 0

- When 0 - IRINT is set to 1 when implicit time integration is used; set to the number of steps in the first step block when explicit time integration is used.
- >0 - Restart file is overwritten every IRINT time steps
- <0 - Restart file is appended every IRINT time steps

Mass Formulation

Allows you to choose the type of mass matrix to be used in dynamic analysis. Creates the MASSTYP field on the NXSTRAT entry. Choose Consistent (enters 0 for MASSTYP) or Lumped (enters 1 for MASSTYP). Default = 0

Analysis Options

Large Strain Form - Indicates which large strain formulation is used for 4-node shell elements. Creates the ULFORM filed on the NSXSTRAT entry. There are three choices:

- 0..Auto (ULFORM = 0, default) - Uses the Updated Lagrangian-Jaumann (ULJ) formulation when the Rigid-target algorithm for contact is used in SOL 601 or when SOL 701 is run. Otherwise, the Updated Lagrangian-Hencky (ULH) formulation is used.
- 1..Updated Lagrangian-Jaumann - Forces use of the ULJ formulation.
- 2..Updated Lagrangian-Hencky - Forces use of the ULH formulation

Shell Thickness Integ - Allows you to choose the integration order for the local t-direction (through thickness) of shell elements. Creates TINT field on the NXSTRAT entry. Choose from Guass Integration with Integration Order from 1 thru 6 or Newton-Cotes integration with order (-3, -5, or -7). (Default = 2..Guass Integration)

Shell DOF Factor - Angle used to determine whether a shell mid-surface node is assigned 5 or 6 degrees of freedom. Creates SDOFANG field on NXSTRAT entry. (Default = 5.0)

Element Death Time Delay - Sets the Element death time delay. Creates the DTDELAY field on the NXSTRAT entry (Default = 0.0). When an element is too deformed and becomes “dead”, its contribution to the overall stiffness of the structure is removed. By specifying DTDELAY > 0.0, the contribution from the element stiffness is gradually reduced to zero over time DTDELAY instead of being removed suddenly. This may help in the convergence of the solution.

Matrix Stabilization Factor - Indicates whether the stiffness matrix stabilization feature is used. When not checked (Default), places a 0 (Matrix Stabilization is not used) in the MSTAB field of the NXSTRAT entry. When checked, places a 1 (Matrix Stabilization is used) in the MSTAB field of the NXSTRAT entry.

When MSTAB = 1, you can also specify a value for the Matrix Stabilization Factor. When a value is specified, FEMAP will write the MSFAC field and the value to the NXSTRAT entry.

u/p Formulation for Almost Incompressible - Indicates whether u/p formulation is used for elements. When not checked (Default), creates a 0 (u/p formulation is not used) with the UPFORM field on the NXSTRAT entry. When checked, places a 1 (u/p formulation is used instead of displacement-based formulation) with the UPFORM field on the NXSTRAT entry.

Note: u/p formulation is always used for hyperelastic elements and always NOT used for elastic elements with Poisson's ratio less than 0.48. It is also not used for gasket elements. (Gasket elements can not be created in FEMAP at this time).

Displacements Applied to Deformed - Indicates whether prescribed displacements are applied to the original configuration or the deformed configuration. When not checked (Default), creates a 0 (Applied to original configuration) with the DISPOPT field on the NXSTRAT entry. When checked, places a 1 (Applied to deformed configuration) with the DISPOPT field on the NXSTRAT entry.

Note: This option is only applicable for a restart analysis or when a delay (or arrival) time is specified for the prescribed displacement.

Loads Change with Deformation - Indicates whether prescribed loads (pressure and centrifugal) are deformation-dependent (direction and magnitude of the load may change due to large deformation of the structure. When not checked (Default), creates a 0 (Load is independent of structural deformation) with the LOADOPT field on the NXSTRAT entry. When checked, places a 1 (Load is affected by structural deformation) with the LOADOPT field on the NXSTRAT entry.

Note: This option is only applicable for large displacement analysis (PARAM,LGDISP,1 in NX Nastran).

Incompatible Modes for 4 Node Shells - When checked (ICMODE = 1), incompatible modes are used for 4-node shell elements. Creates the ICMODE field on the NXSTRAT entry (Default = 1 for SOL 601, 0 for SOL 701).

Use NXN v8.5 Elastic Beam Formulation - When checked (BEAMALG = 1), the algorithm for elastic beam formulation from NX Nastran 8.5 is used instead of the current algorithm for elastic beam formulation.

Max Disp/Iteration - Specifies a limit for the maximum incremental displacement allowed for any node in any equilibrium iteration. Creates the MAXDISP field on the NXSTRAT entry (Default = 0.0, which means there is no limit on displacement).

Note: Limiting the displacement is generally useful for contact analysis where rigid body motion exists.

Drilling DOF Factor - For shell nodes where the drilling stiffness is zero, this factor is multiplied by the maximum rotational stiffness at the node and assigned as the drilling stiffness. Creates the DRILLKF field on the NXSTRAT entry and the value must be a real number between 0.0 and 1.0 (Default = 1.0E-4).

Translation Options

9/27-Node Element Conversion - Indicates whether to convert 8-node to 9-node quadrilateral elements and 20-node to 27-node brick elements. When not checked (Default), creates a 0 (No conversion of elements) with the ELCV field on the NXSTRAT entry. When checked, places a 1 (Convert elements as described) with the ELCV field on the NXSTRAT entry.

Note: Also converts 6-node to 7-node triangular elements and 10-node to 11-node tetrahedral elements.

RBAR Opt - Allows you to choose how RBAR elements are handled from a drop-down list. Creates the EQRBAR field on the NXSTRAT entry. Default = 0.

- 0..Small Rigid, Large Flex - RBAR is simulated using rigid option in small displacement analysis and using flexible option in large displacement analysis (PARAM,LGDISP,1)
- 1..Rigid - RBAR is simulated using rigid option (rigid link or constraint equations as determined by the program).

- 2..Flexible - RBAR is simulated using flexible option (spring or beam elements as determined by the program)
- 3..Use Springs - RBAR is simulated by spring elements.

RBE2 Opt - Allows you to choose how RBE2 elements are handled from a drop-down list. Creates the EQRBE2 field on the NXSTRAT entry. Default = 0.

- 0..Small Rigid, Large Flex - RBE2 is simulated using rigid option in small displacement analysis and using flexible option in large displacement analysis (PARAM,LGDISP,1)
- 1..Rigid - RBE2 is simulated using rigid option (rigid link or constraint equations as determined by the program).
- 2..Flexible - RBE2 is simulated using flexible option (spring or beam elements as determined by the program)
- 3..Use Springs - RBE2 is simulated by spring elements.

Note: More information about the handling of RBAR and RBE2 elements can be found in the NX Nastran Theory and Modeling Guide for SOL 601.

Rigid Elem Spring - Stiffness of spring elements that simulate RBAR or RBE2 elements. Creates SPRINGK field and enters value on NXSTRAT entry (Default=0.0). When SPRINGK = 0.0, the program sets the SPRINGK value to SPRINK = EMAX * LMODEL, where EMAX is maximum Young's Modulus of materials in the model and LMODEL is the largest dimension of the model. If no material is specified in the model, EMAX is set to 1.0E12.

Rigid Elem Young's Mod - Young's Modulus of material assigned to beam elements that simulate RBAR or RBE2 elements. Creates BEAME field and enters value on NXSTRAT entry (Default=0.0). When BEAME = 0.0, BEAME is set to EMAX * 100, where EMAX is maximum Young's Modulus of materials in the model. If no material is specified in the model, EMAX is set to 1.0E12.

Rigid Elem Effective Area - Circular Cross-Section area of beam elements that simulate RBAR or RBE2 elements. Creates BEAMA field and enters value on NXSTRAT entry (Default=0.0). When BEAMA = 0.0, the program automatically sets the BEAMA = $(\text{LMODEL} * 0.01)^2$, where LMODEL = largest dimension in the model.

Rigid Elem Critical Length - Critical length for determining how RBAR or RBE2 elements are simulated when the rigid or flexible option is used to simulate RBAR, i.e, when EQRBAR = 1 (Rigid) or 2 (Flexible) and/or EQRBE2 = 1 (Rigid) or 2 (Flexible). Creates RBLCRIT field and enters value on NXSTRAT entry (Default=0.0). If RBLCRIT = 0.0, then:

- if EQRBAR (or EQRBE2) = 1, RBLCRIT = LMODEL * 1.0E-6
- if EQRBAR (or EQRBE2) = 2, RBLCRIT = LMODEL * 1.0E-3

LMODEL = largest dimension in the model

Other Parameters

Bolt Force Increments - Number of steps for applying the bolt pre-load force. Creates BOLTSTP field on NXSTRAT entry. This can be used to apply bolt pre-loads incrementally if the solution fails to converge when the total pre-load force is applied in one step. (Default=1)

Num Subgroups - Number of sub-groups to divide large number of elements with same property ID into. Creates NSUBGRP field on NXSTRAT entry. Normally, elements with the same type and property ID are placed into a group. If a group contains more than 1000 elements and NSUBGRP>1, the elements are placed into NSUBGRP sub-groups for more efficient processing. (Default=1)

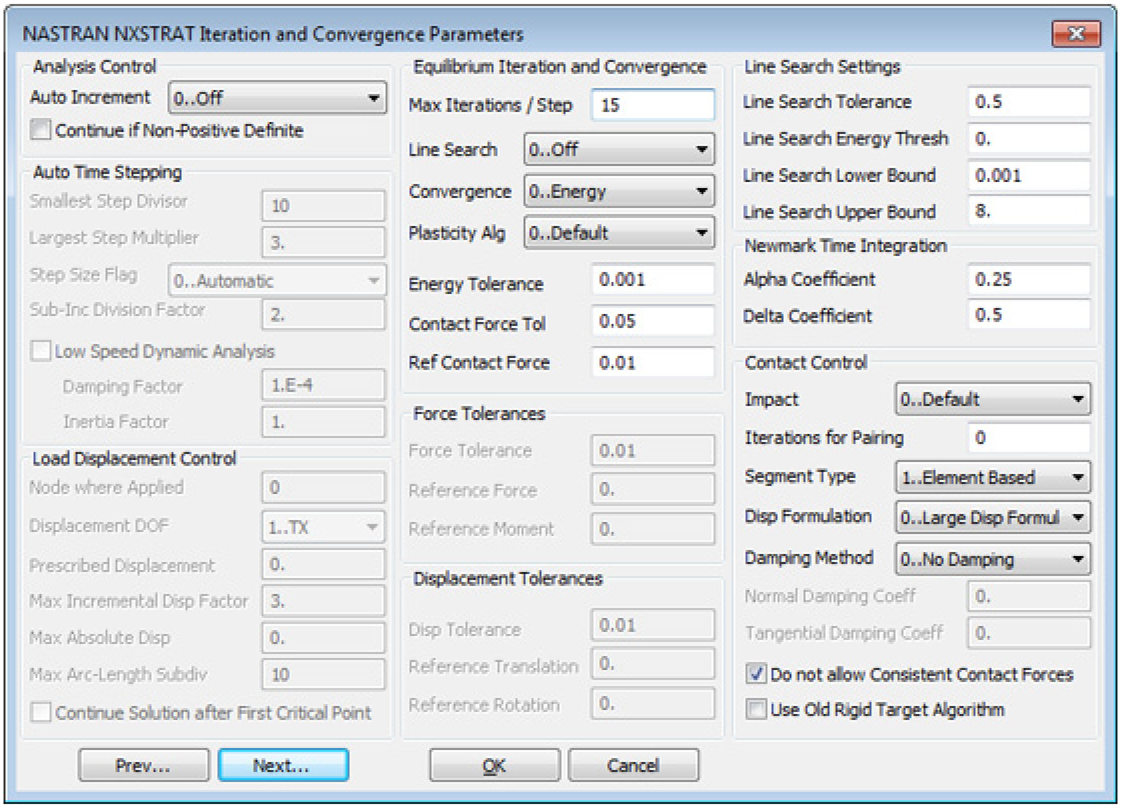
Convert Dependency to True Stress - Checking this option indicates the values in the TABLES1 entry (created using *Model, Function* in FEMAP) will be converted from engineering stress-strain to true stress strain. Creates CVSSVAL field on NXSTRAT entry. When unchecked, the stress is not converted. (Default=0, unchecked)

Allow Element Rupture - Checking this option indicates the table in the TABLES1 entry (created using *Model, Function* in FEMAP) will NOT be extended by linear extrapolation of the two last points, which may be used to allow element rupture at the last specified strain value. Creates XTCURVE field on NXSTRAT entry. When unchecked, the table is extended (Default=1, unchecked)

Solid Results in Material CSys - Checking this option indicates the material coordinate system will be used for output of nonlinear 3D element stress/strain results. Creates ELRESCS field on NXSTRAT entry. When unchecked, the results are output in element coordinate system. (Nastran Default=0, but the default for FEMAP=1, checked)

NASTRAN NXSTRAT Iteration and Convergence Parameters

This box allows you to set up Iteration and Convergence Parameters for SOL 601 (NX Nastran Only).



Analysis Control

Auto Increment - Allows you to choose whether an automatic incrementing scheme is enabled. Creates the AUTO field on the NXSTRAT entry. You can choose from:

0..Off - No automatic incrementing scheme is used

1..On - Automatic time stepping (ATS) scheme is enabled

2..Load-Displacement - Automatic load-displacement control (LDC) scheme is used.

3..Total Load - Total load application (TLA) scheme is enabled. NX Nastran ignores any time step and time function specified. Instead, 50 time steps of 0.2 are used with a linear ramp time function (i.e., 100% of load at time = 10.0) and parameters are set as follows:

Max Iterations/ Step (MAXITE) = 30

Smallest Step Divisor (ATSSUBD) = 64

Line Search is turned "on" (LSEARCH=1)

MAXSDISP = 0.05*(maximum model dimension).

4..Total Load, Stabilize - Total load application with stabilization (TLA-S) scheme is enabled. In addition to TLA, stabilization is used. The following values are set in addition to the ones set when using TLA:

Matrix Stabilization Factor in *NXSTRAT Solver Parameters* dialog box turned "on" (MSTAB=1) and corresponding value (MSFAC) set to 1.0E-10.

Low Speed Dyn Damp Factor turned "on" (ATSLOWS=1) and value set by NX Nastran.

Default = 0 (*0..Off*)

Continue of Non-Positive Definite - Indicates whether analysis continues when the system matrix is not positive definite. When not checked, creates a 0 (Analysis may stop) with the NPOSIT field on the NXSTRAT entry. When checked, creates a 1 (Analysis continues) with the NPOSIT field on the NXSTRAT entry.

Note: If NPOSIT = 0, analysis stops unless

- the ATS or LDC scheme is enabled (AUTO = 1 or AUTO =2 on NXSTRAT)
- contact analysis is being performed

It is not recommended to set NPOSIT = 1 for a linear analysis.

Auto Time Stepping (ATS scheme, AUTO = 1 only)

Smallest Step Divisor - Number that limits the smallest time step size when the automatic time stepping (ATS) scheme is used. Enters the value with the ATSSUBD field on the NXSTRAT entry. For a time step size of DT (on TSTEP entry), the program will stop if convergence is not achieved and the next subdivided time step size is less than DT/ATSSUBD (Value must be greater or equal to 1; Default = 10).

Largest Step Multiplier - Factor that limits the maximum time step size when the automatic time stepping (ATS) scheme is used. Enters the value with the ATSMXDT field on the NXSTRAT entry. The ATS scheme may increase the time step size after convergence is achieved. However, for a time step size of DT (on TSTEP entry), the program will not use a time step greater than ATSMXDT * DT (Default =3.0)

Step Size Flag - Allows you to choose what time step to use once convergence is reached after an ATS subdivision from a drop down menu. Creates the ATSNEXT field on the NXSTRAT entry.

- 0..Automatic - Automatically set by program. For contact analysis, ATSNEXT = 2, otherwise ATSNEXT = 1.
- 1..Match Convergence- Use the time step size that achieved convergence (the reduced time step size that led to convergence is used again)
- 2..Original - Returns to the original time step size (original time step size before any subdivisions took place).
- 3..Match Solution Time - Uses a time step size such that the solution time matches the original solution time specified by the user.

Sub-Inc Division Factor - Division factor used to calculate the sub-increment time step size. Places value with ATSDFAC field on the NXSTRAT entry. If current time step size is DT (on TSTEP entry) and convergence is not achieved, the next time step size will be DT/ATSDFAC. (Default = 2.0)

Low Speed Dyn Damp Factor - Indicates whether a low-speed dynamics analysis is performed instead of a static analysis. When not checked (Default), places a 0 (Low-speed dynamics option is not activated) with the ATSLOWS field on the NXSTRAT entry. When checked, places a 1 (Low-speed dynamics is performed) with the ATSLOWS field on the NXSTRAT entry.

When low speed dynamic analysis is activated:

Damping Factor - creates the ATSDAMP field on the NXSTRAT entry. Default = 1.0E-4.

Inertia Factor - creates the ATSMASS field on the NXSTRAT entry. Default = 1.0.

Load-Displacement Control (LDC scheme, AUTO = 2 only)

Node where Applied - Grid point ID at which a displacement is prescribed for the first solution step. Creates the LDCGRID field on the NXSTRAT entry. When used, must have an integer value greater than 0.

Displacement DOF - Allows you to choose degree of freedom for prescribed displacement at grid point from the drop-down menu. Enters integer from 1 to 6 with LDCDOF field on NXSTRAT entry. DOF integers:1 = X translation, 2 = Y Translation, 3 = Z translation, 4 = X rotation, 5 = Y rotation, and 6 = Z rotation.

Prescribed Displacement - Magnitude of prescribed displacement at grid point used by LDCGRID for first solution step. Places value with LDCDISP field on NXSTRAT entry.

Max Incremental Disp Factor - Displacement convergence factor used to limit the maximum incremental displacement during a solution step. Enters value with LDCIMAX field on NXSTRAT entry. (Default = 3.0)

Max Absolute Disp - Maximum (absolute magnitude) displacement (for DOF specified by LDCDOF) at grid point LDCGRID allowed during the analysis. Enters value with LDCDMAX field on NXSTRAT entry. When the displacement reaches or exceeds LDCDMAX, the program will stop the analysis.

Max Arc-Length Subdiv - Maximum number of arc length subdivisions allowed. Places value with LDCSUBD field on NXSTRAT entry. Value must be an integer greater than or equal to 1; Default = 10.

Terminate after First Critical Point - Indicates whether the solution will be terminated when the first critical point on the equilibrium path is reached. When not checked (Default), places a 0 (Solution stops) with the LDC-CONT field on the NXSTRAT entry. When checked, places a 1 (Solution continues) with the LDCCONT field on the NXSTRAT entry.

Equilibrium Iteration and Convergence

Max Iterations / Step - Maximum number of iterations within a time step. Enters the value with the MAXITE field on th NXSTRAT entry. If the maximum number of iterations is reached without achieving convergence (see Convergence Criteria, CONVCRI section for more details), the program will stop unless the automatic time stepping (ATS, AUTO = 1) or Load-displacement control (LDC, AUTO = 2) scheme is selected. Value must be an integer between 1 and 999; Default = 15).

Line Search - Allows you to choose the use of line searches within the iteration scheme. Creates the LSEARCH field on the NXSTRAT entry. (Default = 0). The line search options are:

- -1..Off - Line Search is not used.
- 0..Automatic - Automatically set by program.
- 1..On - Line search used.

Convergence - Allows you to choose convergence criteria from a drop-down list. Creates the CONVCRI field on the NXSTRAT entry. Default = 0

- 0..Energy - Convergence based on energy.
- 1..Energy and Force - Convergence based on energy and force.
- 2..Energy and Displacement - Convergence based on energy and displacement.
- 3..Force - Convergence based on force.
- 4..Displacement - Convergence based on displacement.

Plasticity Alg - Allows you to choose a plasticity algorithm from a drop-down list. Creates the PLASALG field on the NXSTRAT entry. Default = 1

- 0..Default - Field not written, currently, “Algorithm 1” is default.
- 1..Algorithm 1 - “Algorithm 1” is used. If this algorithm does not allow convergence, try “Algorithm 2”.
- 2..Algorithm 2 - “Algorithm 2” is used. This algorithm has additional convergence checks.

Energy Tolerance - Relative energy tolerance. Enters value with ETOL field on NXSTRAT entry. Default = 0.001. Used for convergence criteria 0..Energy, 1..Energy and Force, or 2..Energy and Displacement (CONVCRI = 0, 1, or 2)

Contact Force Tol - Relative contact force tolerance. Enters value with RCTOL field on NXSTRAT entry. Default = 0.05. Used for ALL convergence criteria.

Ref Contact Force - Reference contact force. Enters value with RCONSM field on NXSTRAT entry. Default = 0.01. Used for ALL convergence criteria.

Force Tolerances (CONVCRI = 1 or CONVCRI = 3)

Force Tolerance - Relative force (and moment) tolerance. Enters value with RTOL field on NXSTRAT entry. Default = 0.01

Reference Force - Reference Force. Enters value with RNORM field on NXSTRAT entry.

Reference Moment - Reference Moment. Enters value with RMNORM field on NXSTRAT entry.

Displacement Tolerances (CONVCRI = 2 or CONVCRI = 4)

Disp Tolerance - Relative displacement (translation and rotation) tolerance. Enters value with DTOL field on NXSTRAT entry. Default = 0.01

Reference Translation - Reference translation. Enters value with DNORM field on NXSTRAT entry.

Reference Rotation - Reference rotation. Enters value with DMNORM field on NXSTRAT entry.

Line Search Settings (LSEARCH = 0 or LSEARCH = 1)

Line Search Tolerance - Line Search convergence tolerances. Enters value with STOL field on NXSTRAT entry. Default = 0.5

Line Search Energy Thresh - Line Search energy threshold. Enters value with ENLSTH field on NXSTRAT entry. Default = 0.0

Line Search Lower Bound - Lower bound for line search. Enters value with LSLOWER field on NXSTRAT entry. Value must be a real number between 0.0 and 1.0 (Default = 0.001).

Line Search Upper Bound- Upper bound for line search. Enters value with LSUPPER field on NXSTRAT entry. Value must be a real number greater than or equal to 1.0 (Default = 1.0 for contact analysis, 8.0 for analysis with no contact).

Newmark Time Integration

Alpha Coefficient - Alpha coefficient for the Newmark time integration method. Enters value with ALPHA field on the NXSTRAT entry. Default = 0.25

Delta Coefficient - Delta coefficient for the Newmark time integration method. Enters value with DELTA field on the NXSTRAT entry. Default = 0.25

Contact Control

Impact - Allows you to choose the impact control scheme from a drop-down menu. Writes the IMPACT field and value on the NXSTRAT entry (Default = 0). The impact control schemes are:

- 0..Default - No Special treatment is applied for impact problems
- 1..Adjust Vel/Accel - Post impact adjustment of velocities and accelerations is applied
- 2..Mod Newmark Param - Modified parameters are used in Newmark time integration scheme.

Iterations for Pairing - Number of iterations for pairing of contactor node to target segment. Enters value with the NSUPP field on the NXSTRAT entry. If NSUPP>0, during the first NSUPP iterations, the pairing target segment is recorded for each contactor node. From iteration NSUPP+1, if a target segment in the recorded list is repeated, it is “frozen” to be the pairing target segment for the remaining equilibrium iterations in that time step. Specifying NSUPP > 0 may help in the convergence for certain problems. Value must be an integer between 0 and 99; Default = 0.

Segment Type - Selects the type of contact segment to use. Writes the CSTYPE field and value on the NXSTRAT entry (Default = 1). “Element Based” is only applicable when *Contact Type* is set to “0..Constraint Function” on the *NX Adv Nonlin* tab of the *Connection Property*. The segment types are:

- 0..Linear Contact - Use linear contact segment.
- 1..Element Based - Use element-based contact segment which gives better contact traction results.

Disp Formulation - Selects the default displacement formulation used for contact analysis. Writes the CTDISP field and value on the NXSTRAT entry (Default = 0 or 2). The Formulations are:

- 0 or 2..Large Disp Formulation (default) - Use large displacement formulation (contact conditions are updated)
- 1..Small Disp Formulation – Use small displacement formulation (contact conditions are not updated)

CTDISP is a global option since it applies to all contact definitions in the model. If you would like to prevent/allow a specific contact set from updating, the DISP option on the BCTPARA bulk entry can be used (See Section 4.4.3.2, "NX Nastran Contact Property Options - Advanced Nonlinear Analysis (NX Adv Nonlin tab)".

A different formulation may be selected for each individual contact set via BCTPARA entry.

Note: If *Disp Formulation* is set to “1..Small Disp Formulation”, the search of target segments for the contactor nodes is performed only at the beginning of the analysis.

Damping Method - Indicates whether stabilization damping is applied and how it is applied for contact analysis. This feature is generally useful when rigid body motion exists in a model. Writes the CTDAMP field and value on the NXSTRAT entry (Default = 0). The damping methods are:

- 0..No Damping – No stabilization damping is applied

- 1..1st Step Damping – Stabilization damping is applied at the first time step only. The specified damping coefficients (*Normal* and *Tangential Damping*) are applied and ramped down to zero by the end of the first time step
- 2..All Step Damping – The specified stabilization damping coefficients are applied at all time steps.

Normal Damping Coeff - Specifies the normal stabilization damping coefficient. Enters value with CTDAMPN field on the NXSTRAT entry. Default = 0.0

Tangential Damping Coeff - Specifies the tangential stabilization damping coefficient. Enters value with CTDAMPT field on the NXSTRAT entry. Default = 0.0

Do not allow Consistent Contact Forces - Checking this option indicates tensile consistent contact forces will NOT be allowed on quadratic contact segments. Creates TNSLCF field on NXSTRAT entry. When unchecked, the tensile consistent contact forces are allowed. (Nastran Default=1, unchecked)

Use Old Rigid Target Algorithm - Checking this option will direct NX nastran to use the “old” (NX Nastran version 4) rigid contact algorithm. Creates RTALG field on NXSTRAT entry. When unchecked, the “current” rigid target algorithm will be used. (Nastran Default=0, unchecked)

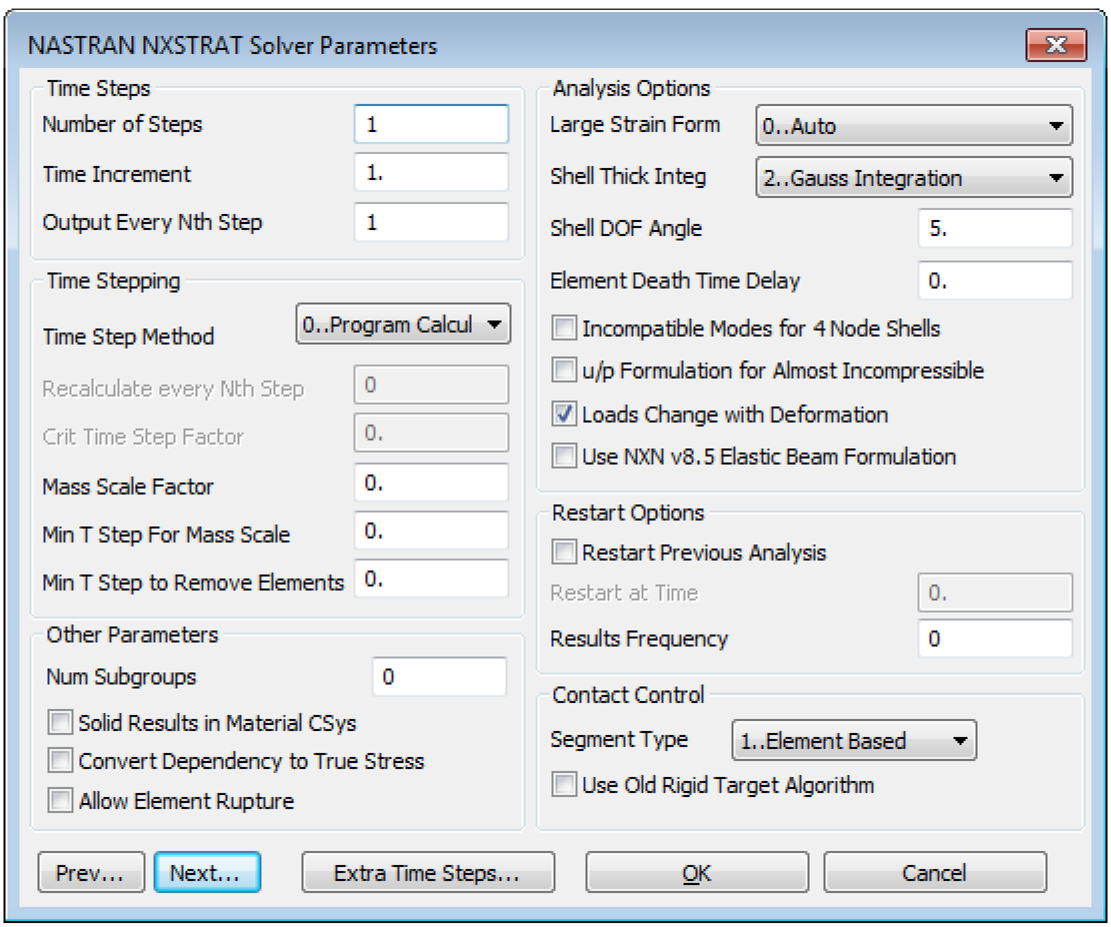
8.7.1.25 Advanced Nonlinear Explicit (NX Nastran Only)

NX Nastran has the ability to perform Advanced Nonlinear Explicit Analysis using Solution Sequence 701 (SOL 701).

A specific Solution 701 dialog box will appear in the Analysis Set Manager when the Analysis Type has been set to either *24..Advanced Nonlinear Explicit*. This dialog box contains Overall Solver Parameters (including Extra Time Step definition) and other solution parameters.

NASTRAN NXSTRAT Solver Parameters

This dialog box allows you to set up Strategy Parameters for SOL 701 (NX Nastran only)



Time Steps

Number of Steps - number of time steps in explicit analysis using SOL 701. Value is entered into the N1 field on the TSTEP entry.

Time Increment - Time increment for steps in transient or nonlinear static analysis using SOL 701. Value is entered into DT1 field on the TSTEP entry.

Output Every Nth Step - Skip factor for output. Creates NO1 field on TSTEP entry. Every NOi-th step will be saved for output.

Time Stepping

Time Step Method - There are two different methods for choosing the time step for explicit analysis. This option creates the XSTEP field on the NXSTRAT entry. The two methods are:

- 0..Program Calculated (XSTEP = 0, default) - Time step is calculated by the program based on the critical time step size. The data in the selected specified in the Time Steps section (TSTEP entry) is used to calculate the total solution time for the analysis.
- 1..User Defined (XSTEP = 1) - The number of time steps and the time step size as specified in the Time Steps section (TSTEP entry) is used.

Recalculate every Nth Step - When this option is set, the critical time step size will be recalculated every Nth step (for example, when N = 2, the critical time step size will be recalculated every 2nd time step). Recalculating the critical time step size often can be computationally expensive, therefore this option is used to decrease solve time when the Time Step Method is set to *1..User Defined*. Creates the XDTCAL field on the NXSTRAT entry.

Crit Time Step Factor - Critical time step size is calculated based on certain assumptions. It is often necessary, especially for nonlinear analysis, to use a time step size smaller than the calculated critical time step size. This factor multiplied by the calculated critical time step size gives the time step size used in the analysis. Only used when Time Step Method is set to *1..User Defined.* Creates XDTFAC field on NXSTRAT entry. (Default = 0.9)

Mass Scale Factor - Specifies the factor to scale the mass (densities) of the entire model (at the beginning of the analysis) to increase the critical time step size required for stability when the explicit time integration scheme is used. Creates the XMSCALE field on the NXSTRAT entry. (Default = 1.0)

Min Step For Mass Scale - Minimum time step size used to determine if mass scaling will be applied to elements (at the beginning of analysis) whose critical time step size is smaller than the given value (XDTMIN1). Creates XDTMIN1 field on NXSTRAT entry. The amount of mass scaling is calculated for each element so that the critical time step size is equal to XDTMIN1.

Min Step to Remove Elements - Minimum time step size used to determine whether an element will be removed in an explicit time integration analysis. In explicit time integration, the smaller the element size, the smaller the critical time step size will be for the analysis. If the critical time step size for an element is smaller than XDTMIN2, the element will be removed in the analysis. Creates XDTMIN2 field on NXSTRAT entry.

Other Parameters

Num Subgroups - Number of sub-groups to divide a large number of elements with the same property ID into. Creates the NSUBGRP field on the NXSTRAT entry. Normally, elements with same type and property ID are placed into a group. If a group contains more than 1000 elements and the Num Subgroups > 1, the elements are placed into that number of subgroups for more efficient processing.

Solid Results in Material CSys - Checking this option indicates the material coordinate system will be used for output of nonlinear 3D element stress/strain results. Creates ELRESCS field on NXSTRAT entry. When unchecked, the results are output in element coordinate system. (Nastran Default=0, but the default for FEMAP=1, checked)

Convert Dependency to True Stress - Checking this option indicates the values in the TABLES1 entry (created using *Model, Function* in FEMAP) will be converted from engineering stress-strain to true stress strain. Creates CVSSVAL field on NXSTRAT entry. When unchecked, the stress is not converted. (Default=0, unchecked)

Allow Element Rupture - Checking this option indicates the table in the TABLES1 entry (created using *Model, Function* in FEMAP) will NOT be extended by linear extrapolation of the two last points, which may be used to allow element rupture at the last specified strain value. Creates XTCURVE field on NXSTRAT entry. When unchecked, the table is extended (Default=1, unchecked)

Analysis Options

Large Strain Form - Indicates which large strain formulation is used for 4-node shell elements. Creates the ULFORM filed on the NSXSTRAT entry. There are three choices:

- 0..Auto (ULFORM = 0, default) - Uses the Updated Lagrangian-Jaumann (ULJ) formulation when the Rigid-target algorithm for contact is used in SOL 601 or when SOL 701 is run. Otherwise, the Updated Lagrangian-Hencky (ULH) formulation is used.
- 1..Updated Lagrangian-Jaumann - Forces use of the ULJ formulation.
- 2..Updated Lagrangian-Hencky - Forces use of the ULH formulation

Shell Thickness Integ - Allows you to choose the integration order for the local t-direction (through thickness) of shell elements. Creates TINT field on the NXSTRAT entry. Choose from Guass Integration with Integration Order from 1 thru 6 or Newton-Cotes integration with order (-3, -5, or -7). (Default = 2..Guass Integration)

Shell DOF Angle - Angle used to determine whether a shell mid-surface node is assigned 5 or 6 degrees of freedom. Creates SDOFANG field on NXSTRAT entry. (Default = 5.0)

Element Death Time Delay - Sets the Element death time delay. Creates the DTDELAY field on the NXSTRAT entry (Default = 0.0). When an element is too deformed and becomes “dead”, its contribution to the overall stiffness of the structure is removed. By specifying DTDELAY > 0.0, the contribution from the element stiffness is gradually reduced to zero over time DTDELAY instead of being removed suddenly. This may help in the convergence of the solution.

Incompatible Modes for 4 Node Shells - When checked (ICMODE = 1), incompatible modes are used for 4-node shell elements. Creates the ICMODE field on the NXSTRAT entry (Default = 1 for SOL 601, 0 for SOL 701).

u/p Formulation for Almost Incompressible - Indicates whether u/p formulation is used for elements. When not checked (Default), creates a 0 (u/p formulation is not used) with the UPFORM field on the NXSTRAT entry. When checked, places a 1 (u/p formulation is used instead of displacement-based formulation) with the UPFORM field on the NXSTRAT entry.

Note: u/p formulation is always used for hyperelastic elements and always NOT used for elastic elements with Poisson’s ratio less than 0.48. It is also not used for gasket elements. (Gasket elements can not be created in FEMAP at this time).

Loads Change with Deformation - Indicates whether prescribed loads (pressure and centrifugal) are deformation-dependent (direction and magnitude of the load may change due to large deformation of the structure. When not checked (Default), creates a 0 (Load is independent of structural deformation) with the LOADOPT field on the NXSTRAT entry. When checked, places a 1 (Load is affected by structural deformation) with the LOADOPT field on the NXSTRAT entry.

Note: This option is only applicable for large displacement analysis (PARAM,LGDISP,1 in NX Nastran).

Use NXN v8.5 Elastic Beam Formulation - When checked (BEAMALG = 1), the algorithm for elastic beam formulation from NX Nastran 8.5 is used instead of the current algorithm for elastic beam formulation.

Restart Options

Restart Previous Analysis - Indicates mode of Execution. By default this option is not checked, which places a 0 (Normal analysis run, no restarts) in the MODEX field of the NXSTRAT entry. When checked, places a 1 (Restart Analysis) in the MODEX field of the NXSTRAT entry.

Restart at Time - Solution starting time. Creates the TSTART field on the NXSTRAT entry. IF MODEX=1, TSTART must equal a solution time in which data was saved in the previous run. IF TSTART = 0.0, the last time step in the restart will be used.

Results Frequency - Frequency of saving the analysis results in the restart file. Creates IRINT field on the NXSTRAT entry. Default = 0

- When 0 - IRINT is set to 1 when implicit time integration is used; set to the number of steps in the first step block when explicit time integration is used.
- >0 - Restart file is overwritten every IRINT time steps
- <0 - Restart file is appended every IRINT time steps

Contact Control

Segment Type - Selects the type of contact segment to use. Writes the CSTYPE field and value on the NXSTRAT entry (Default = 1). “Element Based” is only applicable when *Contact Type* is set to “0..Constraint Function” or “1..Penalty Method” on the *NX Explicit* tab of the *Connection Property*. The segment types are:

- 0..Linear Contact - Use linear contact segment.
- 1..Element Based - Use element-based contact segment which gives better contact traction results.

Use Old Rigid Target Algorithm - Checking this option will direct NX nastran to use the “old” (NX Nastran version 4) rigid contact algorithm. Creates RTALG field on NXSTRAT entry. When unchecked, the “current” rigid target algorithm will be used. (Nastran Default=0, unchecked)

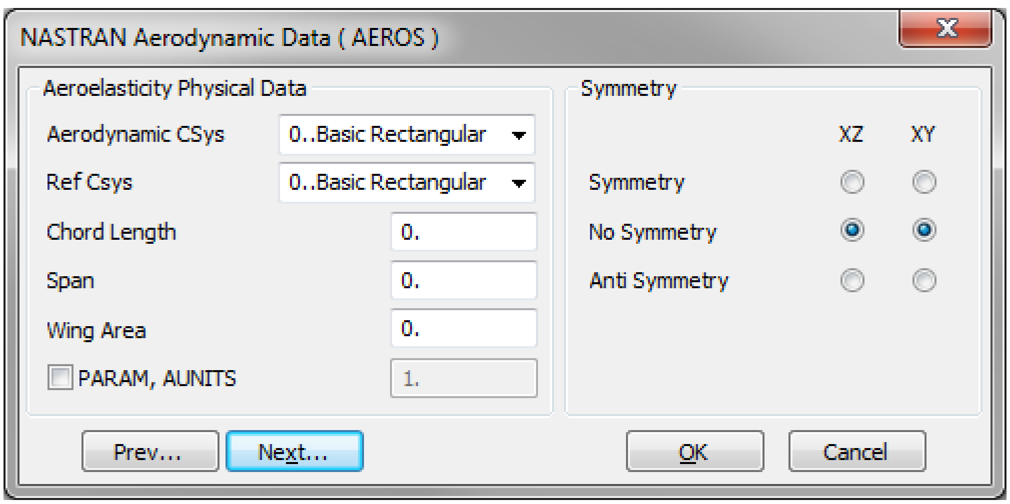
8.7.1.26 Static Aeroelasticity Analysis

NX and MSC/MD Nastran have the ability to perform Static Aeroelasticity analysis using Solution Sequence 144 (SOL 144).

Specific Solution 144 dialog boxes will appear in the Analysis Set Manager when the *Analysis Type* has been set to *25..Static Aeroelasticity*. The *NASTRAN Aerodynamic Data (AEROS*) dialog box allows you to enter basic parameters for static aeroelasticity and an optional conversion factor PARAM used for all subcases. On the other hand, the *NASTRAN Aeroelastic Trim Parameters* dialog box contains a number of “Trim Parameters”, which may be specified in the “Master Requests and Conditions” for an analysis with no subcases or specified individually for each subcase.

AEROF and APRES will be written to case control to request results from static aeroelastic analysis.

NASTRAN Aerodynamic Data (AEROS)



Aerodynamic Physical Data

Aerodynamic CSys - specifies the aerodynamic coordinate system. Must be a rectangular coordinate system. Flow is in the +X direction. Value written to the ACSID field of the AEROS entry.

Ref CSys - specifies the reference coordinate system. Must be a rectangular coordinate system. All AESTAT degrees-of-freedom defining trim variables will be defined in this coordinate system. Value written to the RCSID field of the AEROS entry.

Chord Length - specifies reference chord length. Value written to the REFC field of the AEROS entry.

Span - specifies reference span. Value written to the REFB field of the AEROS entry.

Wing Area - specifies reference wing area. Value written to the REFS field of the AEROS entry.

PARAM, AUNITS - writes PARAM, AUNITS to the Nastran input file with the specified value. This parameter is used to convert accelerations specified in units of gravity on the TRIM entries to units of distance per time squared.

Symmetry

XZ - specifies the symmetry “key” for the x-z plane of the *Aerodynamic CSys*. Based on option selected for *XZ*, writes an integer to the SYMXZ (*Symmetry* = +1, *No Symmetry* = 0, *Anti-Symmetry* = -1).

XY - specifies the symmetry "key" for the x-y plane of the *Aerodynamic CSys*, which can be used to simulate "ground effects". Based on option selected for *XY*, writes an integer to the SYMXY (*Symmetry* = -1, *No Symmetry* = 0, *Anti-Symmetry* = +1).

NASTRAN Aeroelastic Trim Parameters

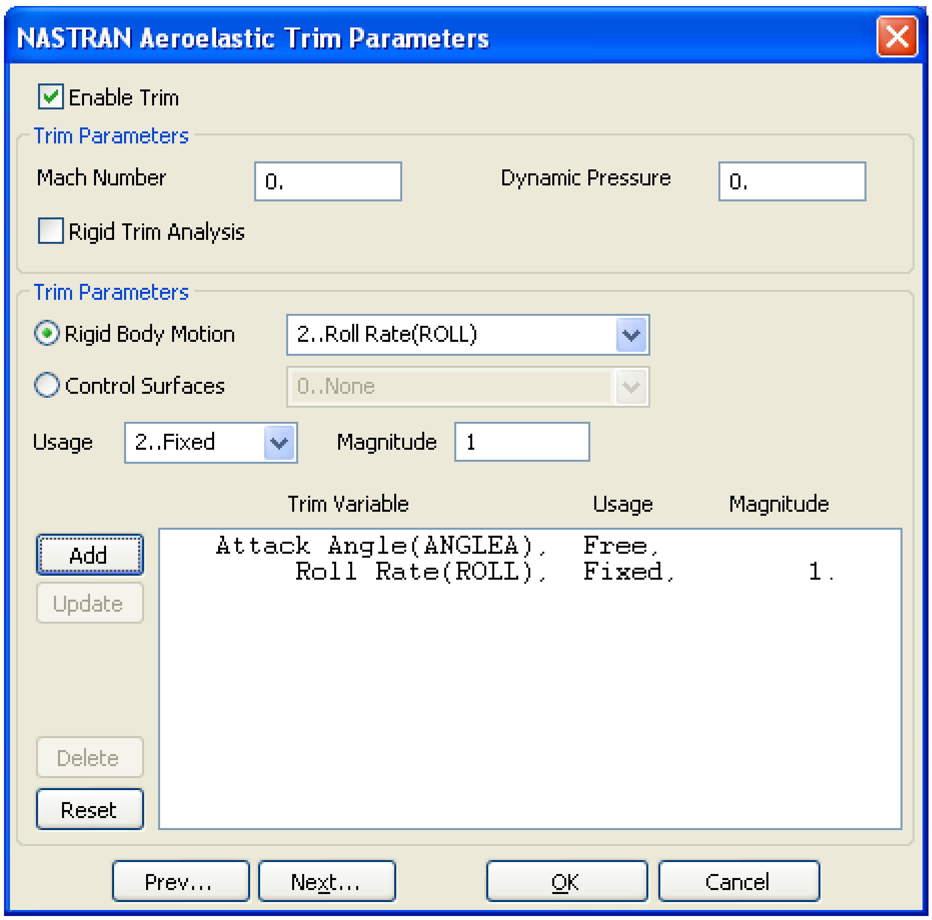
The *Enable Trim* check box may be used to toggle the options set in the *NASTRAN Aeroelastic Trim Parameters* dialog on/off in the Master case and for each subcase.

The *Trim Parameters* in the upper portion of the dialog box are used to define values on the TRIM bulk data entry.

Mach Number - specifies the mach number. Value written to the MACH field of the TRIM entry.

Dynamic Pressure - specifies a value for dynamic pressure. Value written to the Q field of the TRIM entry.

Rigid Trim Analysis - specifies if trim analysis is rigid. When "on" a value of 0.0 is written to the AEQR field of the TRIM entry. When "off", a value of 1.0 is written to the AEQR field of the TRIM entry.



The *Trim Parameters* in the lower portion of the dialog box write AESTAT and/or TRIM entries using values entered for various "Trim Variables" in the list.

When set to *Rigid Body Motion*:

a. Select from the list of "Standard Labels Defining Rigid Body Motions" on the AESTAT (ANGLEA, SIDES, ROLL, PITCH, YAW, URDD1, URDD2, URDD3, URDD4, URDD5, and URDD6)

b. Select a *Usage* (1..Free or 2..Fixed). If 2..Fixed, enter a magnitude as well (UXi value on TRIM entry).

c. Click *Add* to add the "Trim Variable" to the list in the lower portion of the dialog box.

When set to *Control Surfaces*:

a. Select from the list of *Aero Control Surfaces* in your model, then follow steps b and c above.

To update a “Trim Variable”, highlight one in the list, set the appropriate values, then click *Update*. The *Delete* button is used to delete a single highlighted “Trim Variable” from the list, while *Reset* will delete all “Trim Variables” from the list.

8.7.1.27 Aerodynamic Flutter Analysis

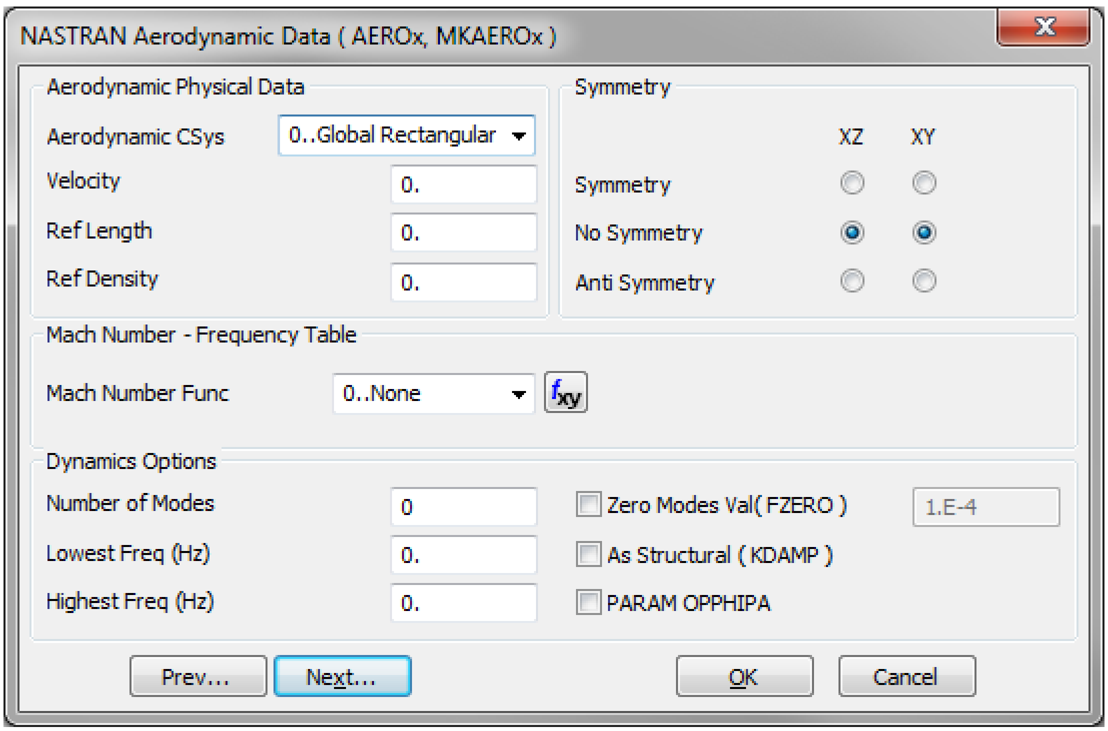
NX and MSC/MD Nastran have the ability to perform Aerodynamic Flutter analysis using Solution Sequence 145 (SOL 145).

Specific Solution 145 dialog boxes will appear in the Analysis Set Manager when the *Analysis Type* has been set to *26..Aerodynamic Flutter*. The *NASTRAN Aerodynamic Data (AEROx, MKAEROx*) dialog box allows you to enter basic parameters for unsteady aerodynamics, a table of Mach numbers vs. Reduced frequencies, and some additional dynamic analysis information. On the other hand, the *NASTRAN Flutter Parameters* dialog box contains a number of “Flutter Parameters”, which may be specified in the “Master Requests and Conditions” for an analysis with no subcases or specified individually for each subcase.

The standard *NASTRAN Modal Analysis* dialog box is also used to setup a Flutter analysis. See Section 8.7.1.9, "NASTRAN Modal Analysis" for more information about the options available in this dialog box.

When using the PK method, results from the Flutter Summery Table will be imported into FEMAP as functions.

NASTRAN Aerodynamic Data (AEROx, MKAEROx)



Aerodynamic Physical Data

Aerodynamic CSys - specifies the aerodynamic coordinate system. Must be a rectangular coordinate system. Flow is in the +X direction. Value written to the ACSID field of the AERO entry.

Velocity - specifies the velocity for aerodynamic force data recovery and to calculate the BOV parameter. Value written to the VELOCITY field of the AERO entry.

Ref Length - specifies reference length for reduced frequency. Value written to the REFC field of the AERO entry.

Ref Density - specifies reference density. Value written to the RHOREF field of the AERO entry.

Symmetry

XZ - specifies the symmetry “key” for the x-z plane of the *Aerodynamic CSys*. Based on option selected for *XZ*, writes an integer to the SYMXZ (*Symmetry* = +1, *No Symmetry* = 0, *Anti-Symmetry* = -1).

XY - specifies the symmetry “key” for the x-y plane of the *Aerodynamic CSys*, which can be used to simulate “ground effects”. Based on option selected for *XY*, writes an integer to the SYMXY (*Symmetry* = -1, *No Symmetry* = 0, *Anti-Symmetry* = +1).

Mach Number - Frequency Table

Select a function to specify a list of Mach Numbers vs. Reduced Frequencies (Type of function MUST be "34..Mach Number vs. Freq"). To create a new function "on-the-fly", click the N*ew Function* icon button. Writes as many MKAERO2 entries as needed for all XY data pairs in the function (4 data pairs per MKAERO2).

Dynamics Options

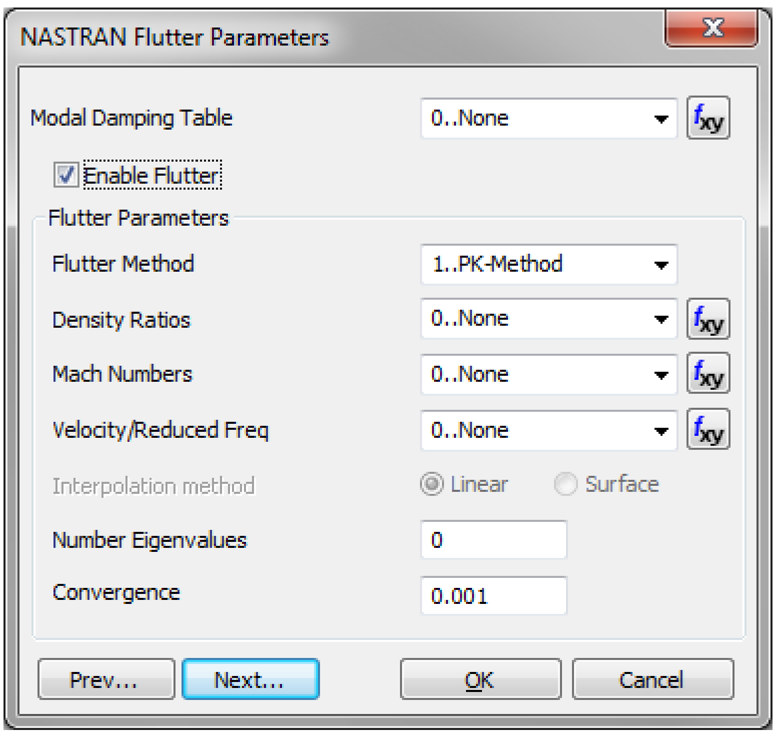
These options allow you to limit the modes used to analyze the response of the structure by allowing you to set a subset of the frequency range specified in the *NASTRAN Modal Analysis* dialog box or simply enter a fewer number of modes to use. This can be useful if restarting from a *Modal Analysis* which had a larger frequency range or more modes than are needed to run an accurate *Modal Transient* analysis.

Number of Modes will write the PARAM,LMODES entry, *Lowest Freq (Hz)* will write PARAM,LFREQ and *Highest Freq (Hz)* will write PARAM,HFREQ. Specify *Rigid Body Zero Modes (FZERO)* to have modes with values under specified value be considered "0".

When checked, *As Structural(KDAMP)* will write out PARAM,KDAMP,-1, which causes the viscous modal damping, specified by the *Modal Damping Table* in the *NASTRAN Flutter Parameters*, to be entered into the complex stiffness matrix as structural damping. When checked, PARAM OPPHIPA will write out PARAM,OPPHIPA,1, which will output the real vibration modes at all degrees of freedom, including the aerodynamic degrees of freedom.

NASTRAN Flutter Parameters

The *Model Damping Table* can be specified here (function Type must be "6..Structural Damping vs. Freq", "7..Critical Damping vs. Freq", or "8..Q Damping vs. Frequency") and writes a TABDMP1 entry..



The *Enable Flutter* check box may be used to toggle the options set in the *NASTRAN Flutter Parameters* dialog on/off in the Master case and for each subcase. A FMETHOD= # case control entry will be written to each subcase, specifying which FLUTTER entry to use for each subcase.

Flutter Parameters

Flutter Method - specifies the flutter analysis method. There are four methods available:

0..K-Method (K written to METHOD field on FLUTTER entry)

1..PK-Method (PK written to METHOD filed on FLUTTER entry). Is the default method.

2..PKNL-Method (PKNL written to METHOD on FLUTTER entry). Is PK-Method with no looping.

3..KE-Method (KE written to METHOD on FLUTTER entry). Is K-Method restricted for efficiency.

Density Ratios - select a function to specify the density ratio vs. aerodynamic factor. Type of function must be “35..vs.Aerodynamic Factor”. Function values written to FLFACT entry which is then referenced by the DENS field of the FLUTTER entry.

Mach Numbers - select a function to specify the mach numbers vs. aerodynamic factor. Type of function must be “35..vs.Aerodynamic Factor”. Function values written to FLFACT entry which is then referenced by the MACH field of the FLUTTER entry.

Velocity/Reduced Freq - select a function to specify the velocity (PK and PKNL methods) or reduced frequencies (K and KE methods) vs. aerodynamic factor. Type of function must be “35..vs.Aerodynamic Factor”. Function values written to FLFACT entry which is then referenced by the RFREQ field of the FLUTTER entry.

Interpolation Method (K and KE methods only) - specify an interpolation method for aerodynamic matrix interpolation. Choose between *Linear* (writes L to IMETH field on FLUTTER entry. Default) or *Surface* (writes S to IMETH field on FLUTTER entry).

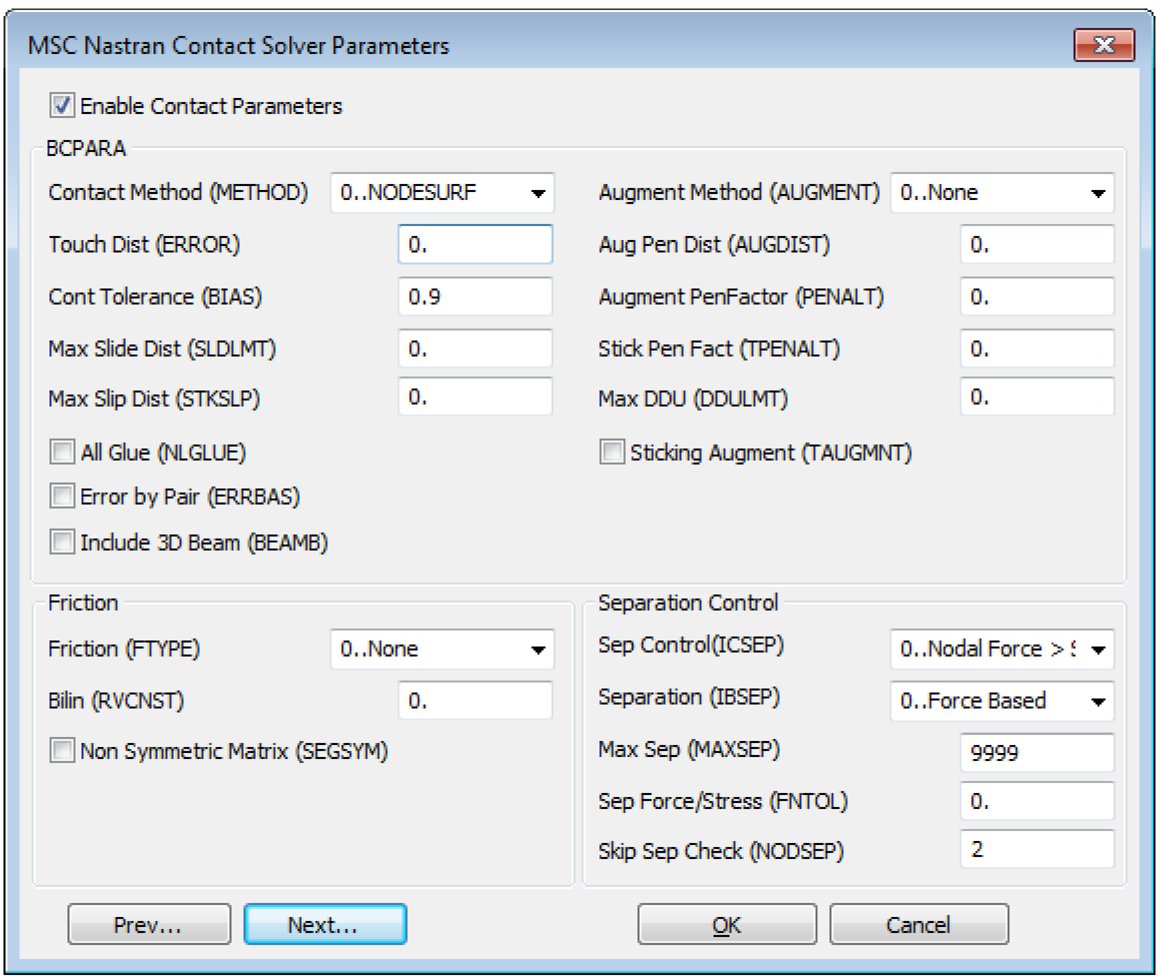
Number Eigenvalues (PK and PKNL methods only) - specify the number of eigenvalues. Value written to NVALUE field on the FLUTTER entry.

Convergence (PK and PKNL methods only) - specify a convergence value for k, which a value used to accept eigenvalues. Value written to EPS field on the FLUTTER entry.

8.7.1.28 Contact Parameters (MSC Nastran Only)

The *MSC Nastran Contact Solver Parameters* dialog box contains a number of options which can be specified when performing an analysis in MSC Nastran, which also includes surface-to-surface contact. All of the items in this dialog box correspond to items (Params) which can be included on the BCPARA entry. For more information, see the BCPARA (Contact Parameters - SOLs 101 and 400) entry in the MSC Nastran Quick Reference Guide.

You must check the “Enable Contact Parameters” box for FEMAP to use any options in this dialog box:



8.7.1.29 Superelement Analysis

NX and MSC/MD Nastran have a number of Superelement Solution Sequences. In fact, all the SOLs in the 100 range are actually Superelement Solutions Sequences (For example, SOL 101 for Linear Static Analysis).

While not accessed through the standard "path" of the *Analysis Set Manager* (i.e., using the *Next* and/or *Previous* buttons), a number of *Analysis Types* contain *External Superelement Creation* and *External Superelement Reference* options which may be accessed.

External Superelement Creation

Located in the *Master Requests and Conditions* branch, the *External Superelement Creation* entry offers the ability to setup an "External Superelement Creation" analysis run. The upper portion of this dialog box is used to specify the EXTSEOUT Case Control command, while the *DOF Sets* section offers an alternate place to choose a constraint set used to write out the ASET or the QSET. Finally, the *Entity ID Range Checks* section simply offers a "check" for duplicate Node and Element IDs, but does not actually write anything to Nastran. For more information, see Section 5.10, "External Superelements Modeling".

External Superelement Reference

Located in the *Options* branch, the *External Superelement Reference* entry offers the ability to setup an "External Superelement Assembly" analysis run. A Nastran Superelement license is required for this type of analysis. The *External Superelement Reference* dialog box contains a number of icons used to create, edit, or delete a reference to a *.pch file, a *.op2 file + *.asm file combination, or a *.op4 file + *.asm file combination. In addition, *PARAM,SECOMB* maybe toggled on/off, and so can *Write Full Path*. When off, only the name of the selected *.asm file will be written to the Nastran input file as an INCLUDE statement. Finally, *Duplicate Node Tolerance* can be toggled on/off and a value provided. When turned on, this writes DPBLKTOL=*value* System Cell to the NASTRAN line of the input file. For more information, see Section 5.10, "External Superelements Modeling".

8.7.2 Writing a Nastran Model with File, Export

Note: This method is obsolete and has been removed from the default configuration of FEMAP. If you need to use it for any reason, you must use the *File, Preferences* command, click the *Interfaces* tab, and check the option for "Enable Old Analysis Interfaces". The use of this method for translation is NOT recommended. Use the *Model, Analysis* method instead.

You can write the FEMAP model into a file that can be read by Nastran through either the *Model, Analysis* or the *File, Export, Analysis Model* commands. Although the commands have a different the user interface, they both produce a file that contains the three required sections: Executive Control, Case Control, and Bulk Data.

For all Nastran analysis types, you will define an analysis set, executive and solution options, bulk data options, output requests/boundary conditions.

Starting to Export the Model

Use the *File, Export, Analysis Model* command to export your model to Nastran. Once you select the command, you then select the appropriate version of Nastran.

If you pick *File, Export* after you have already created an analysis set with the *Model, Analysis* command, the software uses the active analysis set to create the file. You won't see any dialog boxes.

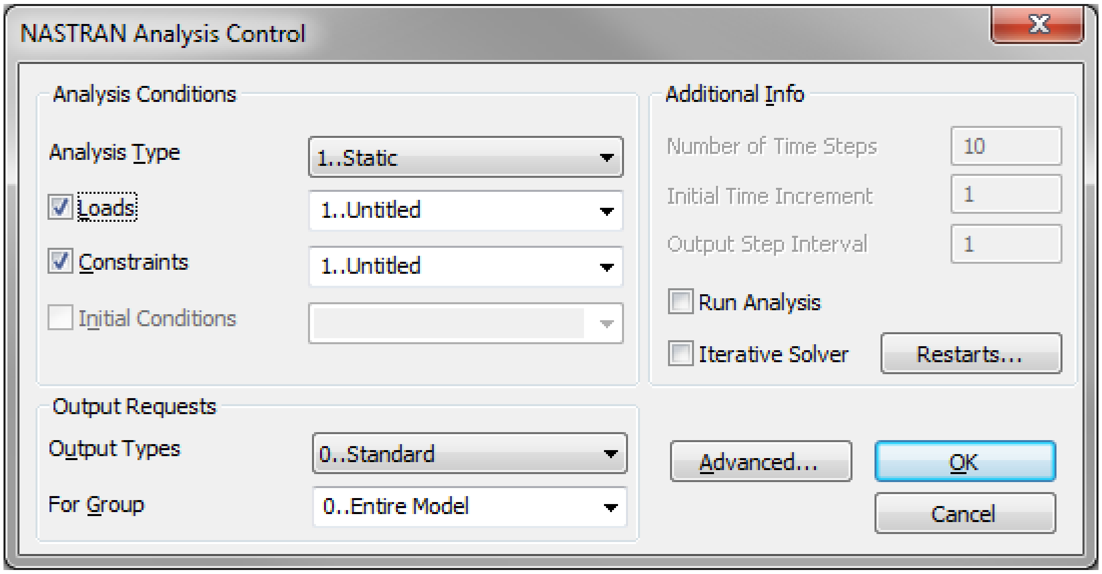
If you don't have an analysis set, you will see the standard file access dialog box where you can choose the name of the file to create. The default filename extension is ".NAS", but you can choose any file name.

The remainder of this section describes the dialog boxes for the interface:

- "Analysis Control"
- "Dynamic Analysis"
- "Heat Transfer Analyses"
- "Bulk Data"
- "Special Notes for Nonlinear, Creep, and Dynamic Analyses"
- "Special Notes for Models with Axisymmetric Elements"
- "Special Notes for Heat Transfer Analysis"

Analysis Control

After you choose the file, you will see the *Nastran Analysis Control* dialog box. Often, this single dialog box is all that is required to create a Nastran input file.



Analysis Type

This list specifies the type of analysis that you want to perform. It should always be set to the same type that you chose in the *File, Export Analysis Model* dialog box, but this gives you an opportunity to change your mind.

Loads, Constraints and Initial Conditions

These lists are used to choose the load and constraint conditions that you want to analyze. Although most analyses require these conditions, if you want to skip them, you can simply turn off the check box to the left of the option that you wish to ignore. Otherwise, you must choose a set from the available lists. The default values will always be the active load/constraint set.

For heat transfer analyses, you will notice that constraint sets are not used. Rather, loads and constraints are both selected from a load set. FEMAP translates nodal temperatures, in the same set as the other thermal loads, as thermal constraints (boundary conditions). In addition however, you can choose a constraint set if you have any constraint equations to be included in your model. Again however, this is only for constraint equations, not nodal constraints

Note: If your analysis requires multiple constraint sets, you will have to press Advanced and go through a more detailed definition of the case control. This dialog box can only be used for a single constraint set analysis. This is also true for multiple load sets except for static and nonlinear static analysis types. If multiple load sets are required for these two types, you can still go through *Advanced,* or press the *Loads* button at the bottom right portion of the box. This option allows you to select FEMAP load sets with the standard *Entity Selection* dialog box. The load sets will be written as separate subcases in the order in which they were chosen. Remember, if you use this option, each load set to be analyzed will have the same constraint set.

Output Types

This list allows you to request the type of output you want. The default - *Standard Output* - varies depending on the type of analysis that you are performing. For example, for static analyses it includes displacements, constraint forces, applied loads, elemental forces and elemental stresses, while for modal analysis it includes only displacements and constraint forces. The list also includes other specific options (*Displacements Only*, *Displacements and Stresses*) and an *All* option. The *All* option does not necessarily request all output available in Nastran. It does request the same output as standard plus some additional types like elemental strains and constraint equation forces. If you need more control over output selection, you must press *Advanced*.

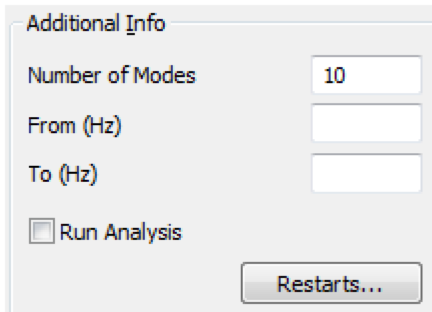
For Group

If you want to limit output to a subset of your model, you can define a group which contains the nodes and elements that you want. When you select that group from this list, all output requests, whether nodal or elemental, will be based on the entities in that group. If you need more control, such as multiple groups, or limiting only certain types of output, you will have to press *Advanced*.

Additional Info

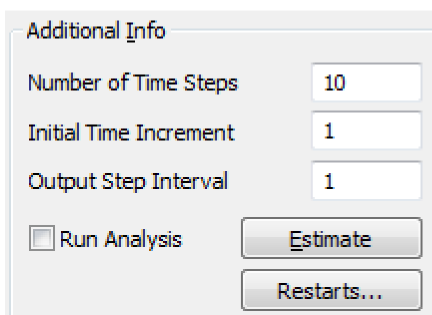
Depending on the type of analysis you are performing, there may be one or more options available in this section that allow you to further control your analysis.

For static, nonlinear static, nonlinear transient response, and steady state heat transfer, no additional info is required or even available. For modal or buckling, you can specify the number of modes/eigenvalues that you want to recover and a frequency range of interest. This is also true for modal transient and modal frequency response analyses.



Heat transfer analyses require control over the number of time steps and the time increment at the start of the analysis. In addition, you can specify an output step interval to minimize the amount of output. For example, an output step interval of 3 means that output will only be written for every third step.

Your choice of the time stepping and time increment are crucial to proper convergence to an accurate solution. To assist you in defining these values, the *Estimate* button can be used. to examine the model that you have defined, including material properties, the duration of any functions, and your mesh, and make a guess at the values for the other options. Remember, this is just an educated guess based on your model. It may not be what you intended to analyze. It is ultimately up to you to set these values appropriately.



Run Analysis

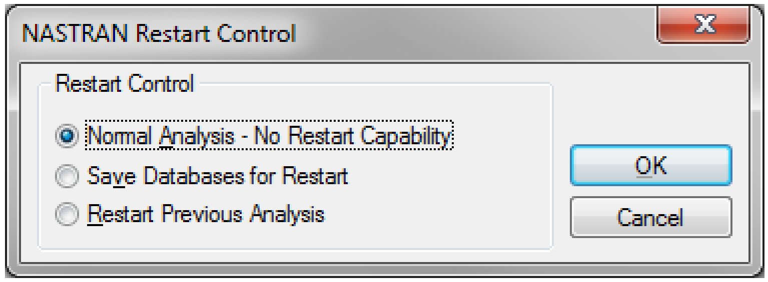
This option is used by the version of FEMAP included in the NX Nastran and MSC.Nastran for Windows product to automatically begin the analysis, otherwise it is not currently used.

Iterative Solver

When this option is checked, FEMAP will write the necessary commands to invoke the Nastran iterative solver.

Restarts...

... allows you to specify whether you want to perform a *Normal Analysis* (the default), *Save Databases for a Restart* (only currently available in the bundled MSC.Nastran for Windows product), or to *Restart from a Previous Analysis*. If you select *Restart Previous Analysis*, the standard file selection dialog box will appear, and you will need to select the old Nastran database from which to restart.



Multiple Load Cases

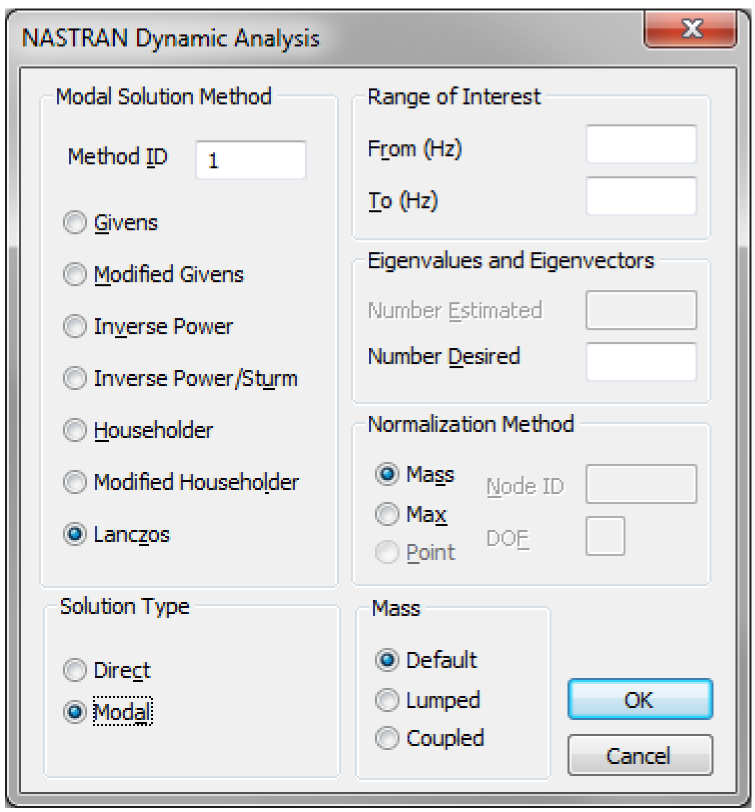
As mentioned previously, this button is only available for static and nonlinear static types and only if your model contains more than one load set. It is used to select multiple load sets to be written as separate subcases. Each subcase will have the same constraint set and print options. If different constraint sets are required, or different print options for subcases are needed, you must use *Advanced*.

Advanced

As mentioned above, if your model is more complicated, or you need more control over the file that is produced, you will have to press the *Advanced* button. This will lead to a series of additional dialog boxes where you can control each step of the translation. The remainder of this section will tell you more about those dialog boxes.

Dynamic Analysis

For normal modes, random, or buckling analyses, these parameters are used to define the EIGR (EIGRL) command that controls your modal analysis. A similar box will appear for transient and frequency response analysis, except you will have the option to select *Direct* or *Modal Solution Type*.



For a description of this dialog box, see Section 8.7.1.9, "NASTRAN Modal Analysis".

Executive Control

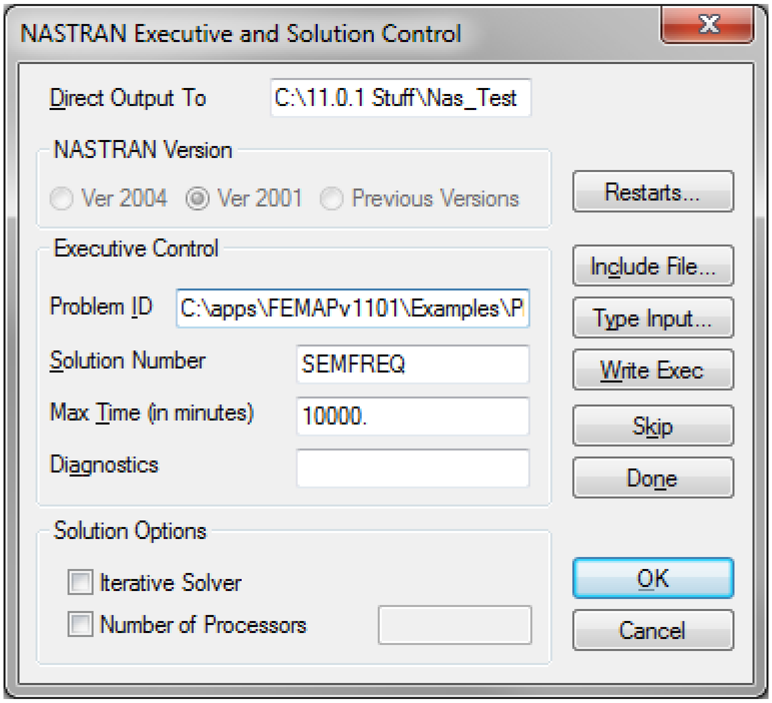
The next dialog is used to define the Executive Control commands for your Nastran model.

The standard Executive Control includes the version number, four executive control commands and two solution options. If you want further customization, the other buttons on the dialog box provide that control

Nastran Version

Select the version that you are using: *Ver 2004* (MSC.Nastran 2004), *Ver 2001* (MSC.Nastran 2001) or *Previous Versions* (up to version 70.7).

Executive Control



•The *Problem ID* is written as a title to the ID command.

•The *Solution Number* selects the DMAP "solution sequence" that will be executed. FEMAP will automatically define this as SESTATICS, SEMODES, SEDTRAN, SEMTRAN, SEDFREQ, SEMFREQ, SEBUCKL, NLSTATIC, NLTRAN, NLSCSH, or NLTCSH, but you can change it to any of the numerical sequences that you want to use.

•The *Max Time* option sets the maximum allowable CPU time for this analysis. Do not set this number too low, or your analysis will terminate prematurely. You can also specify andy diagnostic lines, and whether to use the iterative solver, and set the number of processors. If you simply press *OK*, this standard Executive Control, along with a CEND command will be written.

- The *Direct Output To* option is used by the version of FEMAP included in the MSC.Nastran for Windows product to specify a location for the Nastran output; otherwise, it is not used.

Restarts

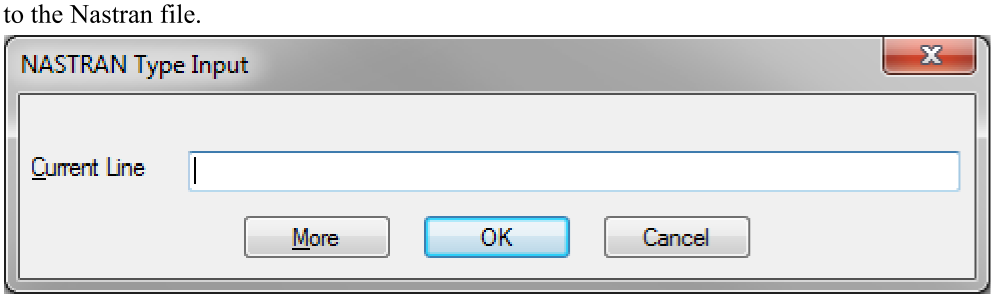
When you press this button, you will see the *Nastran Restart Control* dialog box (shown as "Restarts..." in the *Nastran Analysis Control* dialog box option). You can choose a *Normal Analysis*, *Save Database for Restart*, or *Restart Previous Analysis*. The *Save Database* option is only available in MSC.Nastran for Windows product.

Include File

This button lets you include another text file in the Executive Control section. You can select the file with the standard file access dialog box. This capability can be used to include standard DMAP alter sequences, Job Control (JCL) statements, or other standard modifications to the beginning of your Nastran file. Just save those standard changes in a file, and include them with this option.

Type Input

If you want to make "one-time" changes, press *Type Input,* then enter the line into the following dialog. It will be added to the Nastran file.



After you type your command, press *OK*, or press *More* if you want to continue typing additional commands.

Write Exec and CEND

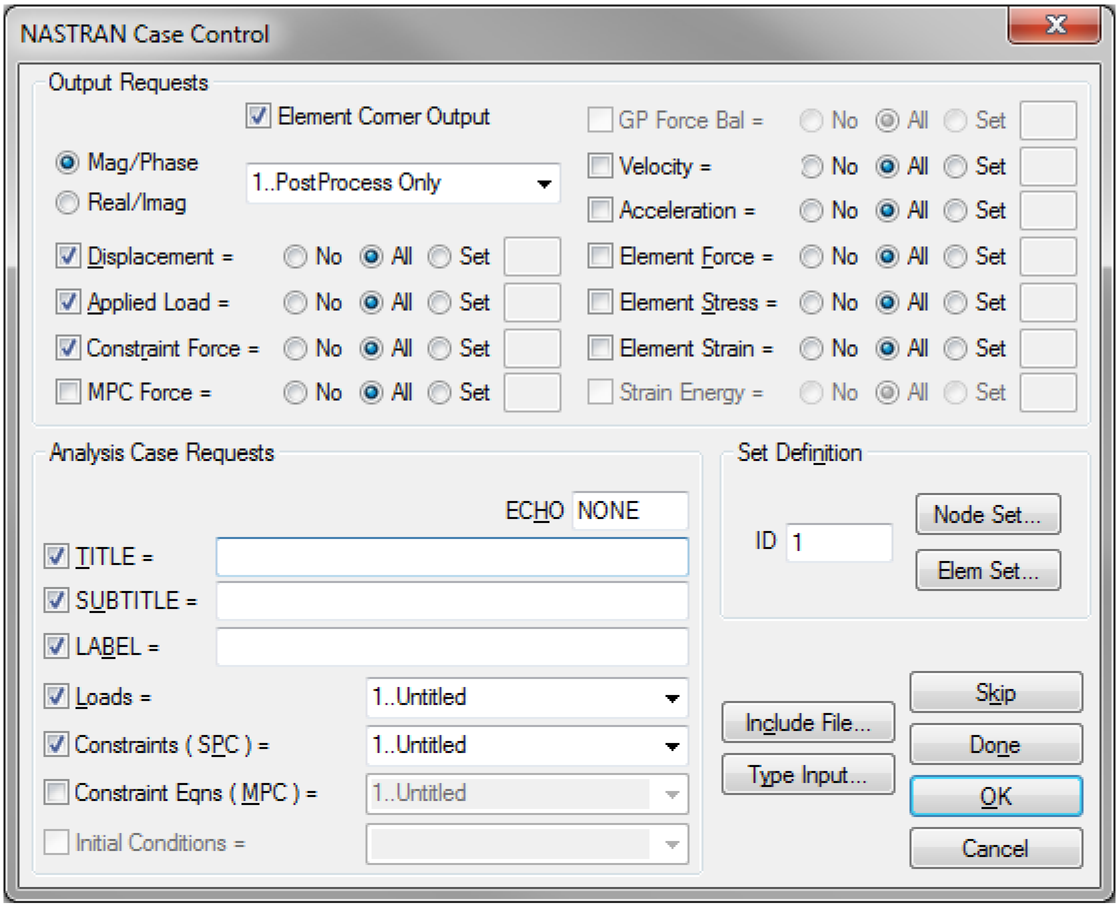
This button writes the standard Executive Control, but stays in the dialog box so you can add additional commands (with *Type* or *Include*) following the standard commands. If you press this button, use the *CEND* button when you want to exit the dialog box. Once you press this button, it and the *OK* button will be disabled to prevent you from accidently writing a second copy of the Executive Control.

Skip

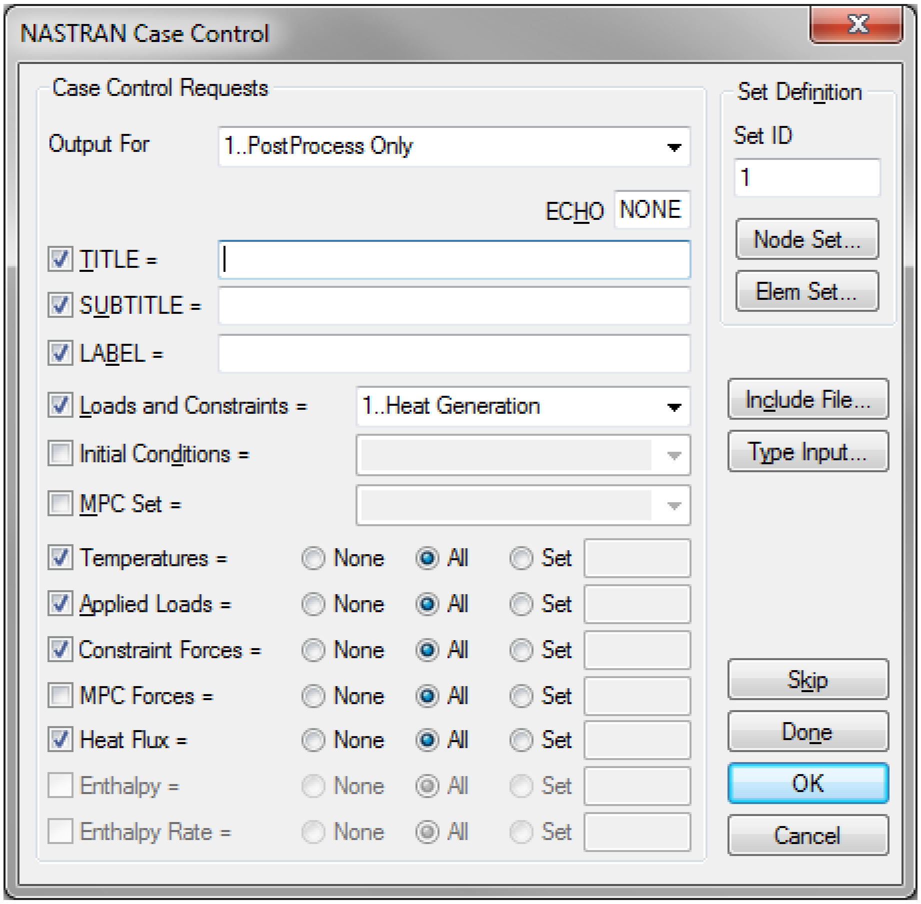
Press this button if you want to go on to the Case Control without writing any additional information.

Case Control

After you have finished the Executive Control, the following dialog box will appear if you are doing a structural analysis:



Or if you are performing a heat transfer analysis, a similar, but slightly different dialog box is used:



While these dialog boxes appear complicated, they are divided into three readily understandable areas: *Set Definition, Analysis Case Requests,* and *Output Requests*. For thermal analysis, both analysis case and output requests areas are combined in the *Case Control Requests* section.

Defining Sets

Usually you want to recover output for all nodes and/or elements, but if you do want to limit your analysis output, you must define a Nastran SET that selects the node or element IDs that you want. You can define these sets with the controls in the *Set Definition* section of the dialog box. First, specify a *Set ID* (which you will refer to later in the *Output Requests* section of the dialog box for output recovery), then push either the *Write Node Set* or *Write Element Set* button. You will see the standard entity selection dialog box, where you choose the entities that you want in the set. The *Set ID* will automatically increment so you can define the next set.

Analysis Case Requests

The standard case control which is written if you simply press *OK* does not use subcases. All selections are written directly to the Master Case. To perform multiple analyses with different load and/or constraint sets, you can define multiple subcases. Subcases are available only for structural non-transient analysis types (i.e. static, normal modes, buckling, and nonlinear static).

Defining Subcases

You can define multiple subcases by choosing a *Subcase ID*, setting the desired options, and pressing the *Write Case* button. The *Subcase ID* will automatically increment, and options for the next subcase can be defined. When you define your last case, press the *OK* button instead of *Write Case* to go to the *Bulk Data* dialog box. If you want some options to apply to all subcases, you can write them to the master case before defining your first subcase by simply pressing *Write Case* before you define a subcase ID.

SUBCASE ID

As stated above, this option should be set to the ID of the subcase you want to create. If it is blank (or 0), the master case will be created.

ECHO

This option determines how the model will be written in the standard output file. The default, SORT, will write your model in sorted alphabetical order. If you do not want the model to be listed in the output change this to NONE.

TITLE, SUBTITLE, and LABEL

These three options specify the titles that will be used at the top of every output "page".

LOAD, SPC and MPC Sets and Initial Conditions

These options choose the loads, constraints, and initial conditions for this case. FEMAP will write all available constraint sets in the Bulk Data portion of the file. FEMAP will also write all available load sets in the Bulk Data section except for heat transfer (steady state and transient), and structural transient solution types (transient response, frequency response, and nonlinear transient). Only load sets which are chosen in the Case Control will be written in the Bulk Data for these problems. You must choose the combination of sets that you want to analyze in the Case Control. This is a primary reason for using multiple subcases when this option is available.

The *LOAD* option allows the selection of a load set to activate for this subcase. The *SPC* option chooses the constraint set for nodal constraints, and the *MPC* option chooses the constraint set for constraint equations. If you do not want any of these options, or if you have activated one for all subcases (in the master case), simply turn off the box at the left of these options.

You can also specify initial conditions for the analysis by defining a separate load set which contains only initial conditions, such as displacements or velocities. Initial conditions are only available for transient and heat transfer analyses.

Output Requests

This section controls what output will be calculated and written to the output files. The default output file for this information is the OP2 file, which is chosen with the *PostProcess Only* option. You may change this to the *F06 Option for Print Only*, or select both. If you do not want a certain type of output, switch it to *None*. If you only want output for a selected portion of your model, choose *Set*, and specify the *Set ID*. Remember, however, you must define the set using the *Set Definition* options. If you are going to post-process, you will probably want to choose *All* (or *None,* for those types of output that you do not want). For frequency response analyses, you can also chose to recover output in either magnitude/phase or real/imaginary format.

Include File and Type Input

These buttons allow you to include additional commands in your Case Control. For details on these buttons, see "Case Control" in the Executive Control section.

Write Case and BEGIN BULK

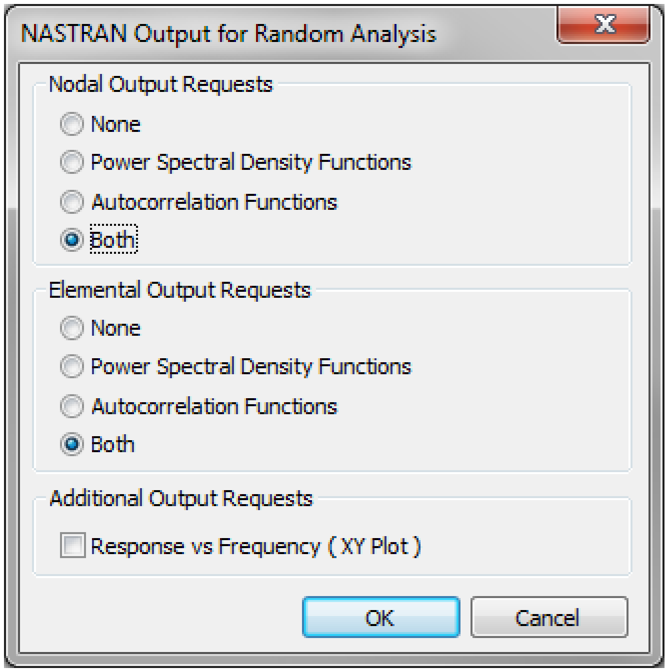
This button writes the selected options to the file and increments the *Subcase ID*. You can then write additional cases, or include custom commands, before proceeding. When you are finished with the Case Control, you must then press BEGIN BULK, to go on to the Bulk Data without writing additional cases. If you press *OK*, you will go on, but the options that you have set will be written as an additional case.

Skip

Press this button if you want to go to the Bulk Data without writing any additional information.

Random Response Analyses

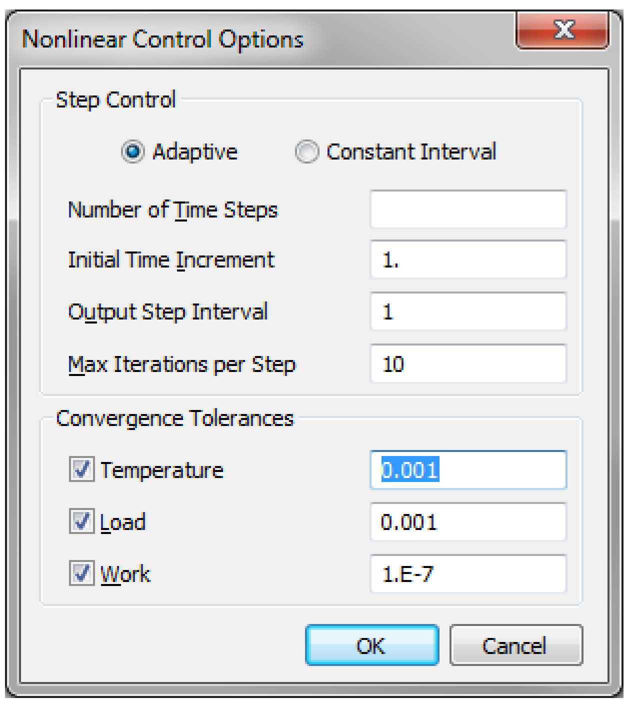
For random response analyses, you must specify additional output requests. The type of output that you request depends on the version of Nastran that you are using.



For a description of this dialog box, see Section 8.7.1.18, "NASTRAN Random Response Analysis".

Heat Transfer Analyses

For heat transfer analysis, you must specify additional iteration control and convergence information.

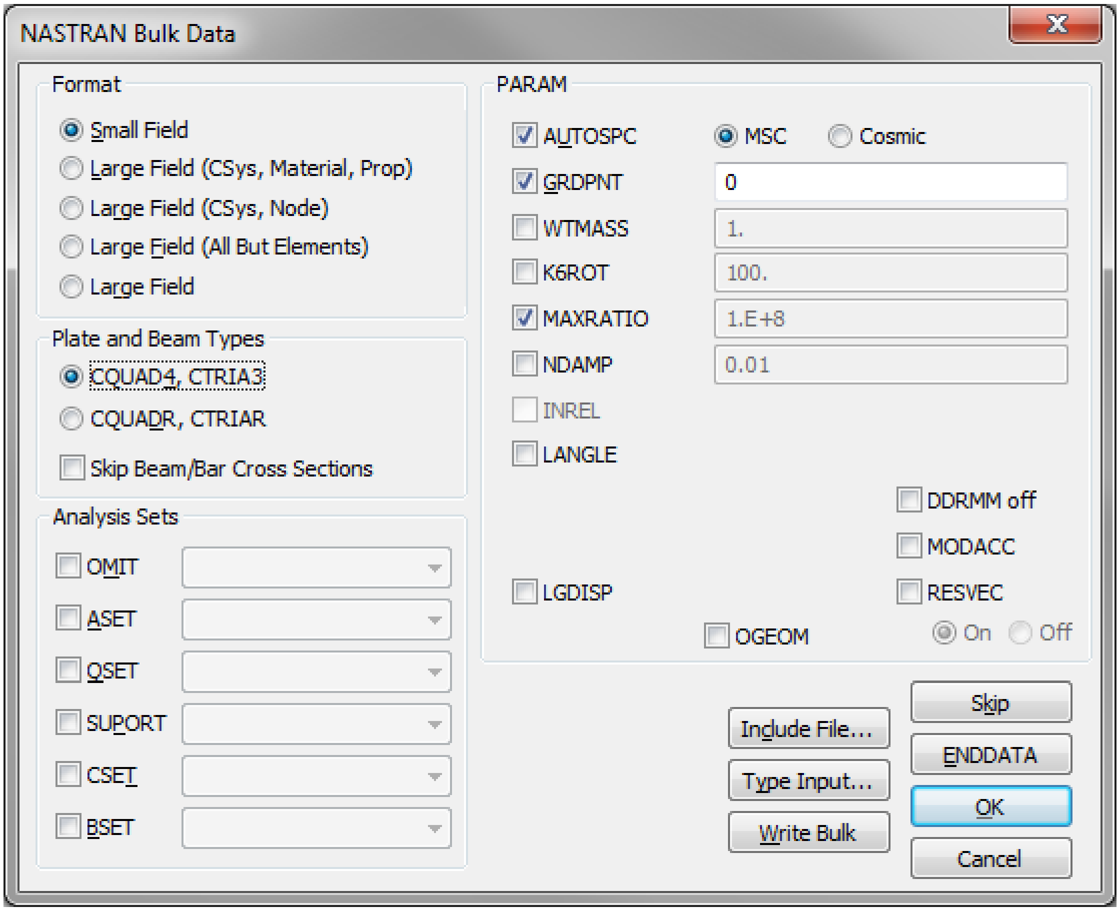


For transient heat transfer, you must input the desired number of time steps, initial time increments, frequency of output and the maximum number of iterations. You can also choose to require convergence on temperature, load, or work criteria within specified values.

For steady state heat transfer, you also have the option of choosing the convergence tolerances, but not the step control.

Bulk Data

After the completing the Case Control, you will see the *NASTRAN Bulk Data* dialog box:



Format

These options determine the format that will be used to write your Bulk Data commands. By default, FEMAP uses small field format (8 character fields). If you want extra precision, for all, or some of your model, you can choose one of the large field formats (16 character fields). The large field formats obviously produce a large file and one that is harder to read. You should not choose that format unless it is necessary. The “limited” large field formats allow you to selectively write large field formats for certain entities and small field format for others. FEMAP does not write free field format.

Plate and Beam Types

Nastran supports two plate formulations:

- The CQUADR and CTRIAR elements have rotational stiffness in the direction normal to the plane of the element.
- By default the CQUAD4 and CTRIA3 elements will be written. These do not have any rotational stiffness in the normal direction.

If you are using a version of Nastran other than NX, MSC or NE, you may want to use the CQUAD4/CTRIA3 option because the CQUADR/CTRIAR elements may not be available.

Nastran can use PBEAM entries or PBEAML entries to define beam properties. You can create both PBEAMs and PBEAMLs in FEMAP using the *Model, Property...* command.

- FEMAP computes values for a “Standard Beam” from the cross-section data supplied and enters the values into the appropriate fields on the *Define Property - BEAM Element Type* dialog box. When a Nastran input deck is exported, FEMAP creates a PBEAM entry for each “standard beam” property defined in the model. Nastran then uses the PBEAM data as it would any other property data to analyze your structure

- When FEMAP creates a “NASTRAN Beam”, the cross-section data supplied is also used to compute values and enters them into the appropriate fields on the *Define Property - BEAM Element Type* dialog box. Upon export to Nastran, FEMAP instead creates PBEAML entries for each “NASTRAN Beam” in the model. PBEAML entries contain cross-section dimension data corresponding to a specific PBEAML Type specified on each PBEAML entry. Nastran uses this cross-sectional data and PBEAML type internally to analyze the structure.

Sometimes you may want to only export PBEAM entries out of FEMAP for analysis purposes. By choosing *Skip Beam/Bar Cross Sections*, FEMAP will use the computed property values from the *Define Property - BEAM Element Type* dialog box and only create PBEAM entries in your Nastran input file, regardless of how the beams were defined.

An example of when this option would be used, would be if you have a model created with “NASTRAN Beams”, which needs to be run by a version of Nastran that does not support PBEAML entries.

PARAM

If you choose these options, FEMAP will write PARAM cards for those options selected. You can also control the format of the AUTOSPC command to the MSC.Nastran convention (PARAM, AUTOSPC, YES) or the Cosmic Nastran convention (PARAM, AUTOSPC, 1). If you want to use additional parameters besides those listed, they can be added with the *Type Input* or *Include File* buttons.

Checking RESVEC allows you to write two forms of the RESVEC PARAM. PARAM,RESVEC,NO which augments static shapes due to applied loads; or PARAM RESVEC,YES which computes residual vectors for applied loads and unit loads (with specified USETi, U6 entries at the desired dofs). When RESVEC is not checked, the PARAM,RESVEC entry will not be written at all, which is required for some types of analysis.

Analysis Sets

These options will write the various mutually exclusive analysis sets to your Bulk Data. Each of these options allows you to select a constraint set that will be translated to the appropriate format for that analysis set.

Typically, for static analysis you will not want to choose any of these sets. For modal analysis however, you will often want to choose an ASET to reduce the number of analysis degrees of freedom. The other sets are rarely used.

Include File and Type Input

These buttons allow you to include additional commands in your Bulk Data. (For information on these buttons, see "Executive Control".)

Write Bulk and ENDDATA

This button writes your model to the Bulk Data portion of the file. You can then use the *Include File* or *Type Input* buttons to include additional information after your model. When you are done, press *ENDDATA* to write the ENDDATA command and exit the translator.

If you want to add data to the file after the end of your model, write the model with the *Write Bulk* button, then add the *ENDDATA* command manually using either *Include File* or *Type Input*. You can then add additional information with these same two buttons. When you are done, press *Skip* to leave the translator without writing an additional *ENDDATA* command.

Skip

Press this button if you want to leave the translator without writing any additional information (including the *ENDDATA* command).

Special Notes for Nonlinear, Creep, and Dynamic Analyses

See Section 8.7.1.21, "Nonlinear Static, Nonlinear Transient, and Creep".

Special Notes for Models with Axisymmetric Elements

See Section 8.7.1.22, "Special Notes for Models with Axisymmetric Elements".

Special Notes for Heat Transfer Analysis

See Section 8.7.1.23, "Special Notes for Heat Transfer Analysis".

8.7.3 Performing a Nastran Analysis

Depending on where you plan to run Nastran, the preliminary steps can be somewhat different. You may not be running Nastran on the same computer as FEMAP, so you may have to transfer the file to the computer where Nastran resides. In some cases, Nastran will also require that your Nastran filename has an extension of ".DAT". In these cases, you will have to either rename the file, or choose a ".DAT" name when you translate (since the FEMAP convention is ".NAS").

Once these things have been completed however, the basic approach to running Nastran is usually the same, simply enter the command:

```
NASTRAN filename
```

where "filename" is the name of the file you created. Again, this can vary depending on the type of computer system you are using. Refer to your Nastran documentation for more information.

If you want FEMAP to launch Nastran and automatically run your analysis, either from the File Analyze command or pressing Analyze in Model Analysis, you must first setup the analysis program. To do this, before running FEMAP, you must establish an environment variable to named MSCNAST_EXE. Set the value of this environment variable to the full path to the Nastran solver executable, for example:

```
SET MSCNAST_EXE = c:\nastran\nastran.exe
```

For details on configuring other solvers to run using local settings See: Section 4.10.2.1, "Run Analysis Using Linked Solver / VisQ / Local Settings" in the *FEMAP Command Guide*

NOTE: Once this environment variable has been defined, FEMAP will be able to launch the analysis program and monitor the job until it is complete. Section 4.10.2.2, "Analysis Monitors"

8.7.4 Reading Nastran Models

Just as you can translate a FEMAP model to Nastran, you can also read Nastran models into FEMAP. When you read a model, FEMAP will read the Executive and Case Control and create a new Analysis Set with the options that were defined in the input file. If a option is not supported by the Analysis Manager then the input will be read as text and inserted into the appropriate Start or End Text fields. In the Bulk Data, FEMAP can read small field, large field, or free format commands.

Limitations

FEMAP cannot currently read some heat transfer specific loads or other commands.

Although support has been added for reading DLOAD, RLOAD2, TLOAD1, NOLINi there is the following limitation:

Each load in the BULK DATA for the dynamics model will be read as a separate load case. You can modify the bulk data so FEMAP will read all of these loads by changing the SID of each load to the same number. However, you will still have to modify these loads in FEMAP because their references to functions, will be lost, although the functions themselves will be read.

FEMAP can directly read most Nastran structural models, but the following conditions are not supported:

- Duplicate element and property IDs cannot be read directly. Some versions of Nastran allow you to use the same element IDs for multiple elements, as long as the elements have different types. FEMAP will read the duplicate information and then ask you if you want to renumber elements, properties, etc. so all entities of a given type (i.e. elements, properties, etc.) have a unique ID.
- MSGMESH commands
- Data replication functions (=, ==, *x, =n, etc.)
- Continuation "cards" that do not directly follow their "parents". This also eliminates the possibility of having multiple parents referencing the same continuation.

If you have any of these conditions in your model, FEMAP will issue errors when you read the file. You can use Nastran to write an expanded, sorted, "punched" data file (use ECHO=PUNCH) which can be read.

In addition to the above limitations, the following conditions are also not supported, but you will have to remove them manually by editing your model.

- Obsolete elements. For more information on the commands that are supported, see Section 7.1, "Translation Table for ANSYS, I-DEAS, NASTRAN, and MSC Patran". If you have other elements, you will have to change them to a supported type before they can be read.

Reading the Model

When you begin to read a file, you will see the standard file access dialog box. Simply choose the file you want and press *OK*. FEMAP will read the file without any further input.

Note: If the Nastran file being imported has INCLUDE files with paths mapped to files on a UNIX drive, a "Drive Map" can be set using the FEMAP.ini file. Any number of "Drive Maps" using this technique.
In the FEMAP.ini file, simply type:
[DriveMap]
/use/"unix_path_to_replace"=(Drive Letter):\"windows_path_to_use"

8.7.5 Post-processing Nastran Output

To post-process Nastran analysis results, you can read the Binary Output 2 results file (typically called "OP2"), the Binary External Data Base results file (typically called "XDB), or the standard printed output file in FEMAP (typically called the "F06" file). There is no special setup, or DMAP, required. FEMAP simply reads the file you would otherwise read manually or print. **In general, when you want to import in the entire results file, you should use the OP2 reader.** It is a little faster than the XDB reader (when reading ALL of the requested results into FEMAP), and much faster and less prone to errors than the F06 reader, although you may want to read the f06 file to obtain the warning and error messages. If you would only like to read in a portion of the results, using the XDB reader may be a better option as FEMAP allows you to pick and choose what to import (See Section 8.7.5.2, "Reading or attaching to the XDB File").

If dealing with a larger output file, it may be better to "attach" to the output file instead of "internalizing" the output data in FEMAP. See Section 2.3.2, "File, Attach to Results" of the *Commands* manual for more information.

Transmitting the File

Since you may have performed the analysis on a different computer, you may have to first transfer the output file to the computer where FEMAP resides. Since this file can be rather large, you may want to "compress" it first, using the NASCRUSH program (described later in this section).

When you transmit the file, you must be certain your communications or networking software does not modify the file in any way. If you are having any problems reading output for post-processing, the first thing to check is the settings in your communication program. If it is expanding carriage control, truncating lines, or modifying the file in any other way, you can have problems. The F06 file must be transmitted as an ASCII file, otherwise carriage returns may not be properly inserted. The OP2 and XDB files should be transferred as a binary file, and FEMAP will perform any byte-swapping that is required when reading the file.

8.7.5.1 Reading or attaching to OP2 and reading FO6 Files

After you run Nastran, choose the *File, Import, Analysis Results* command, select *Nastran* and then *NX Nastran* or *MSC Nastran*. FEMAP will display the standard file access dialog box, and you can choose the output file you want to read. When you press *OK*, FEMAP will immediately open the file and display the first few lines in the *Messages* window. FEMAP will then ask if you really want to read the file you selected. If you answer *Yes*, FEMAP will read your output.

When results are in the .op2 file (or .xdb file), you can also "attach" to the file instead of "internalizing" the file using the *File, Attach to Results* command. See Section 2.3.2, "File, Attach to Results" in the *Commands* manual for more information.

You must always read or attach the Nastran output into a model file with FE entities. If you did not create your model in FEMAP, just use the *File, Import, Analysis Model* command to create a FEMAP model, then read the output into that model.

FEMAP will read many types of Nastran output, including:

DISPLACEMENT	- Displacements/Eigenvectors
VELOCITY	- Velocities
ACCELERATION	- Accelerations
MPCFORCES	- Multipoint Constraint Forces
SPCFORCES	- Constraint Forces
OLOAD	- Applied Loads
FORCE	- Element Forces
STRESS	- Element Stresses (Linear and Nonlinear)
STRAIN	- Element Strains (Linear and Nonlinear)
ESE	- Element Strain Energy
FLUX	- Temperature Gradients and Fluxes
THERMAL	- Temperatures, Heat Flow
ENTHALPY	- Enthalpy
HDOT	- Enthalpy Change

FEMAP will also read the failure indices and strength ratios for layered composite elements and Grid Point Force Balance data.

For dynamic analysis, FEMAP will read in Modal Participation Factors, Modal Effective Mass, Modal Effective Weight, and Modal Effective Mass Fraction functions (XY Output), as well as acceleration and displacement functions (XY Output) for Random Response.

Note: XY Output functions are only read in from the F06 file at this time. The F06 file is created for Error and Warning information when the OP2 is chosen, so these functions often are available for import if you requested them from NASTRAN.

FEMAP will also read complex output, in both magnitude/phase or real/imaginary format, for most of the above structural output types.

In addition to the output in the file, FEMAP can optionally compute additional output during the translation. For example, FEMAP will always compute the magnitude of all displacements and rotations. Likewise, FEMAP will automatically compute principal, max shear, mean, and Von Mises stresses, if they were not read, but the data necessary to compute them is available. For large models with significant amounts of output, you may not want to automatically compute these values. You can skip this computation by turning it off with the *File, Preferences, Interfaces* command. The default is to compute these values.

Naming of Output Sets

By default, FEMAP simply titles Output Sets from Static Analyses sequentially - Case 1, Case 2, etc.. For modal analyses, the mode numbers and frequencies are used, for transient analyses, the time step is included in the title. In all of these cases, the titles may not convey the information that you need to see, so there is an option that allows you to control how titles are created. If you refer to the “Output Set Titles” option of the *File, Preferences, Interfaces* command, there are several options that allow you to use the text that you specify as the TITLE, SUBTITLE or LABEL text in your analysis to be used as the Output Set titles.

Limitations

The following limitations and recommendations apply when you are trying to post-process Nastran output with FEMAP. Most of these limitation apply strictly to the f06 reader, but a few also are pertinent to both readers.

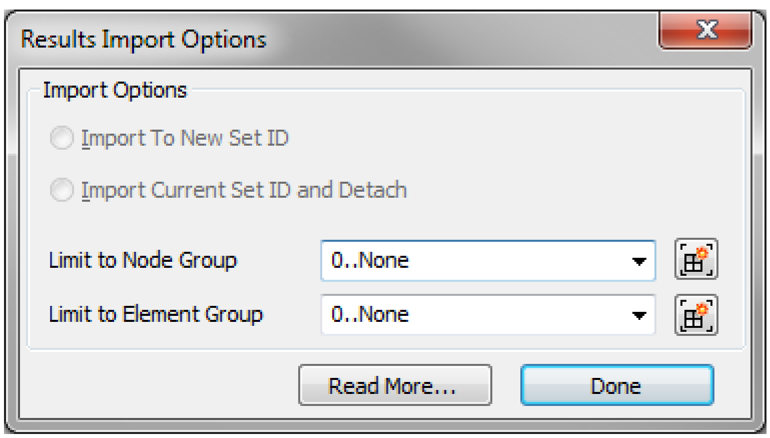
- Elemental corner forces, stresses, and strains are not read for axisymmetric elements. This data can be in your file, but it is skipped.
- For modal analysis, you must request the same output for all eigenvalues which are duplicates for the f06 reader. If you do not, FEMAP may associate output data with the wrong one of the duplicate eigenvalues. Since knowing which eigenvalues are duplicate before the performing the analysis is difficult, it is generally best to request the same output for all modes/eigenvalues. Although it is not required, it is best to always request displacement output in a modal analysis. If you have displacement output, FEMAP will label all of your output sets with the mode number and frequency. Otherwise, the output will be labeled with the eigenvalue.

- For nonlinear static analysis with axisymmetric elements, only the last set of stresses will be read from the f06 file. Therefore, if intermediate step output is contained in your file, FEMAP will create output sets for the different steps, but will only post-process stress results for the last output step.
- Hyperelastic elements return stress and strain data at the Gauss points. FEMAP uses the data at the closest Gauss point as the data at the center or element corner. No interpolation is done. These are really Gauss point values. Corner data is not created for 6-node CPENTA Hyperelastic element data. Gauss points for this element are at element midsides and are not mappable to corner values.
- In general FEMAP requires SORT1 style output for the f06 reader. FEMAP can read some nodal results (displacements, velocities, accelerations, constraint forces, applied loads, temperatures and other heat transfer data) and elemental heat transfer results in SORT2 format if there are less than 2000 steps.

8.7.5.2 Reading or attaching to the XDB File

After you run Nastran, choose the *File, Import, Analysis Results* command, select *Nastran*, and then choose *NX Nastran* or *MSC/MD Nastran* from the drop-down list. FEMAP will display the standard file access dialog box for you to choose the XDB file you want to read. When you press *OK*, FEMAP will immediately open the *Select Output to Internalize* dialog box, which facilitates selection of output sets and vectors. By default, all output sets in the *Output Sets* section will be selected and the *All Output Vectors* option in the *Output Vectors* section will be enabled, thus all output from the XDB file will be imported. To only import a subset of the results from the XDB file, simply select the desired output set(s) and optionally disable the *All Output Vectors* option to be able to select individual output vector(s) for import. For more information on using the *Select Output to Internalize* dialog box see Section 8.6.0.1, "Using the Select Output Sets and Select Results dialog boxes" in the *Commands* manual.

Once you click OK, the *Results Import Options* dialog box will appear:



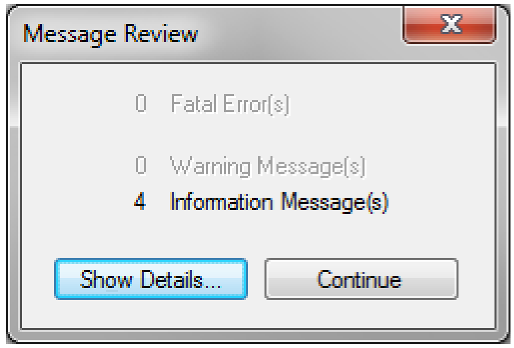
In addition to selecting which output sets and output vectors to internalize, groups limiting output to certain nodes and/or elements may be selected. The “quick group” icons next to the drop-down lists can be used to create a new group or edit an existing group “on-the-fly”. Depending on which button you select, you will be able to choose only nodes or elements. If you click the *Read More...* button, the *Select Output to Internalize* dialog box will be displayed again for selection of additional output set(s) and output vector(s) and this process can be repeated as many times as needed.

Another option is to use the *File, Attach to Results* command. Once an XDB file is attached, you can then use the *Save to Model* functionality of *File, Attach to Results* to ‘internalize’ all or a portion of the results data in the XDB file, much like you can when using File, Import, Analysis Results. See Section 2.3.2, "File, Attach to Results" in the *Commands* manual.

Note: In order for the XDB to be able to read in a certain type of output, you must have requested it BEFORE sending the input file to Nastran to be solved. If the output was not requested, it will not be in the XDB file and therefore not available to be selected for retrieval.

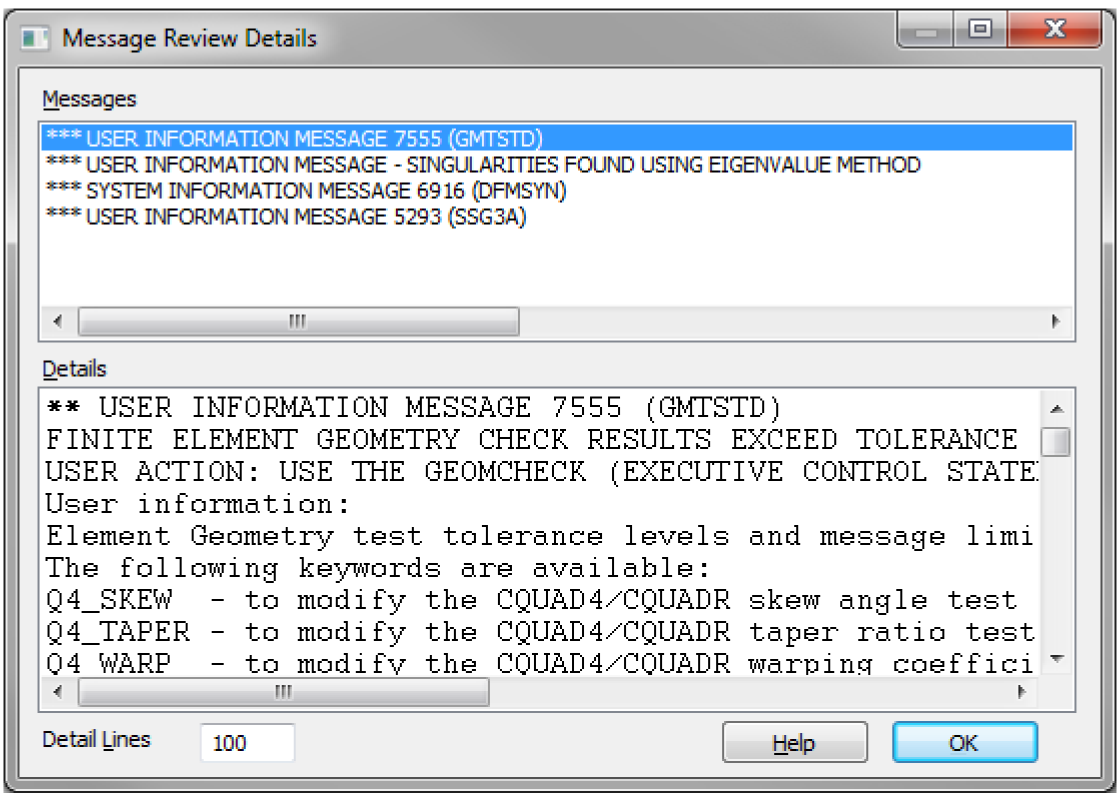
8.7.6 *Reviewing Messages and Errors*

If you are reading the printed (.f06) file, when FEMAP has finished reading the results, you will see the *Message Review* dialog box.



This displays the number of messages of each type found in the file. You will almost always see some warning and information messages. If you want to review the messages (always recommended), press *Show Details* to display the *Message Review Details* dialog box.

The messages will be displayed at the top in the order they appeared in the file. Initially, the *Details* will be blank. Select a message by clicking on it in the top, and the *Details* section will be filled with the number of *Detail Lines* (default is 100 lines to facilitate showing the *Extended Error Messages* from Nastran) from your output file that follow the message. The *Extended Error Messages* are turned on by default



Clicking *Help* will take you directly to a list of Nastran error codes and a brief description of the specific error. This are often helpful in diagnosing what may be causing a Nastran analysis to fail.

Note: It is highly recommended to use the *Extended Error Messages* option when using the Nastran solver. There is then only one screen used to review Nastran Error Messages. If you do not use this option, clicking the *Help* button in *Message Review Details* dialog box will link to an older file of Nastran Error Messages which may or may not be accurate for newer Nastran versions.

The *Message Review Details* dialog box is resizable for FEMAP.

8.8 Patran Interfaces

There are three direct interfaces between FEMAP and MSC.Patran. You can write a FEMAP model to the MSC.Patran format for analysis, read an existing MSC.Patran model, or read analysis results for post-processing. The model interfaces use the MSC.Patran neutral file format. As you can see from the table in "Analysis Software Interfaces", they only support the finite element entities in FEMAP. **You cannot transfer geometry (points, curves, surfaces, or volumes) between FEMAP and MSC.Patran using these interfaces.**

For more information on the entities that are translated, see Section 7.1, "Translation Table for ANSYS, I-DEAS, NASTRAN, and MSC Patran".

8.8.1 Writing a MSC.Patran Model

To translate your model to MSC.Patran neutral format, select *File, Export, Analysis Model* and select *PATRAN* format. You will see the standard file access dialog box with the default extension set to ".PAT". Select the filename that you want and press *OK*. There are no additional options. FEMAP will immediately write the MSC.Patran file.

A Warning About Conventions

MSC.Patran neutral files allow you to write information in "tables" that are later interpreted by the other MSC.Patran neutral file translators. For example, section properties values can be written in any order, but must follow the order required by the neutral file translator that you will select later. We have chosen to follow the conventions required by the MSC.Patran neutral file translators for Nastran (NASPAT and PATNAS). This convention may, or may not, work with other neutral file translators. Check your data carefully.

8.8.2 Reading a MSC.Patran Model

Reading a MSC.Patran file is just as simple as writing one. You will probably want to start a new model before beginning, but then just choose the *File, Import, Analysis Model* command and select *PATRAN* format. As before, you will see the standard file access dialog box. You can simply choose the file that you want to read. You must select a formatted MSC.Patran neutral file. Binary files are not supported.

The only option when reading a MSC.Patran model pertains to constraints. The MSC.Patran neutral file format does not contain nodal constraints, rather they are written as enforced displacement loads with a zero displacement. As you read the model, if FEMAP encounters a zero enforced displacement, you will be asked whether to convert it (and all others) to nodal constraints.

FEMAP versions 9.3 and above read Points, Lines/Curves, Patch/Surface and Named Components/Groups.

The same restrictions on conventions that were described above apply to models that you read. The file must conform to the NASPAT/PATNAS conventions.

8.8.3 Post-processing MSC.Patran Output

FEMAP can read the MSC.Patran Displacement (.DIS), Element Results (.ELS), and Nodal Results (.NOD) files. These files could have come from any analysis program, but if you plan to display the results, you must read them into an equivalent FEMAP model.

To begin reading files, choose the *File, Import, Analysis Results* command and select PATRAN format. You will be asked whether you want to read each type of file. If you answer *Yes*, you will see the file access dialog box and you can choose the file that you want to read. Answer *No* to skip that file and proceed to the next type. Since MSC.Patran writes each output case to a separate file, FEMAP gives you the chance to read additional files after you have read (or skipped) the initial group.

Working with Output From MSC.Patran

Unlike output from other programs, the type of nodal and elemental results in each field of the MSC.Patran neutral file are not defined. Just like the conventions in the model files, the order of output in the file depends on the translator that you used to create it. For this reason, FEMAP simply loads the nodal and elemental output into vectors that are labelled with the MSC.Patran vector numbers. Refer to your MSC.Patran documentation for a description of the data in each vector. You can use the FEMAP *Model, Output, Vector* command to modify the titles after the output has been read.

Since FEMAP does not know what data is contained in the individual vectors, no additional calculations of principal, Max Shear, or Von Mises stress are done when you read MSC.Patran output. Displacement magnitudes are calculated since displacement components are specifically defined in the ".DIS" file.

8.9 Vendor-Supported Interfaces

All configurations of FEMAP include a set of interfaces that are developed and supported by analysis software vendors. This topic briefly describes these interfaces and the general process for using them. For details on working with these analysis programs, see the vendor's user documentation.

8.9.1 Analysis Software Descriptions

The table lists the vendor-supported analysis software interfaces:

Interface	Analysis Software Vendor	FEMAP Writes:	FEMAP Reads:	Notes
CAEFEM	Concurrent Analysis Corp. www.caefem.com	FEMAP neutral file	• CAEFEM results (read automatically if CAEFEM is launched from FEMAP) • FEMAP neutral file (if CAEFEM is run externally)	
NEi Nastran	Noran Engineering, Inc. www.nenastran.com	FEMAP neutral file	FEMAP neutral file FEMAP neutral output file (FNO) NEi Nastran input files	For more information on the FEMAP interface, see Section 8.7, "Nastran Interfaces".
SINDA/G	MSC Software www.mscsoftware.com/Products/CAE-Tools/Sinda.aspx	FEMAP neutral file	FEMAP neutral file	

8.9.2 Using the Interfaces

This topic describes the general process for using the interfaces in the previous section. Once you create the FEMAP model, you:

- write the FEMAP data to a FEMAP neutral file
- launch the analysis software (either from FEMAP or externally)
- restart FEMAP and read in the neutral file containing the analysis results

8.9.2.1 Writing Data from FEMAP to the Analysis Software

Use the *File, Export, Analysis Model* command to write a neutral file to the analysis software. On the *Export To* dialog box, pick the interface, then type of analysis, such as static.

The vendor-supported analysis types listed are:

- CAEFEM
- Nastran. Pick NEi/Nastran.
- SINDA/G. On the SINDA/G Write Options dialog box, choose options to define the case control file.

8.9.2.2 Running the Analysis

Generally, you must exit FEMAP to run the analysis software. In some cases, however, you also have the option of launching the analysis software directly from FEMAP:

- CAEFEM runs as a 32-bit Windows Application. Since it is a true Windows program, you do not have to leave the Windows environment to perform an analysis. You can launch CAEFEM directly from FEMAP.

8.9.3 Reading Analysis Results into FEMAP

If you launch the analysis software from FEMAP, the results will be recovered automatically. If you run the analysis software separately from FEMAP, however, you must restore or restart FEMAP before you can import the neutral file containing analysis results.

The way that you write the analysis results in the analysis software determines how you will import the data into FEMAP for post processing. FEMAP requires that the model information be present before you can import the analysis results.

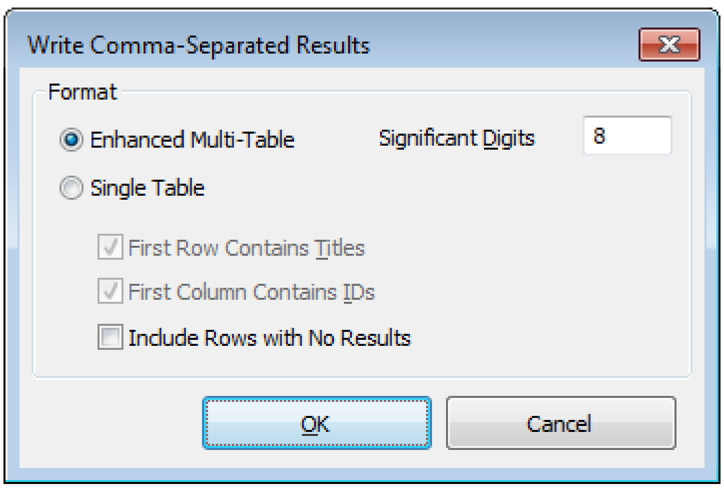
- If you wrote the analysis results into a new file, you will have to load the original model file (*File, Open*) or open a new FEMAP model file and read in the neutral file that was exported to the analysis software (*File, Import, Analysis Model*). Once you have the model, you can import the analysis results (*File, Import, Analysis Results*).
- If you appended the results to the original input file, you can load the model and the results at one time by reading this neutral file *(File, Import, Analysis Results*).

8.10 Comma-Separated Tables

FEMAP contains two translators which allow you to read and write output in a tabular format. This format is easily exchanged between FEMAP and various spreadsheet and database programs, and also provides a simple and fast way to load output into FEMAP from your own programs.

8.10.1 Writing a Comma-Separated Table File

To create a comma-separated table, choose the *File, Export, Analysis Model* command, press the *Other Interfaces* button, select the *Comma-Separated* option in the *Analysis Format* section, then press *OK*. You will see the standard file access dialog box where you can choose the name of the file to create. The default extension is always ".CSV". Once you press *Save*, you will then see the *Write Comma-Separated Results* dialog box, which contains the following options:



Format

These options control how the selected output is formatted.

When using *Enhanced Multi-Table*, the output will be exported using the extended comma-separated table format. This format is much more powerful and robust than the *Single Table* format and should be the format used if you want to output results from multiple output sets and/or plan to import or attach to the results in the CSV file. For more information on this format, see Section 8.10.4, "The Extended Comma-Separated Table Format".

When using *Single Table*, all output will be written to a single CSV table. For more information on this format, see-Section 8.10.3, "The Comma-Separated Table Format"

Note: The *Single Table* format is not optimal, as only nodal or elemental output can be written to a single CSV using this format, not both at the same time. Also, if you are exporting output from multiple output sets, all output will be placed into a single output set if later imported into FEMAP. In addition, you cannot attach to a CSV results file which uses this format.

The *Single Table* format offers several options. If you choose *First Row Contains Titles*, the output vector titles are written at the beginning of the file; otherwise, no titles are given. If you choose *First Column Contains IDs*, the node and element IDs will be written along with the output data. It is usually best to write these IDs so you can relate the output back to the associated nodes or elements. If you choose *Include Rows with No Results*, it simply writes out empty rows.

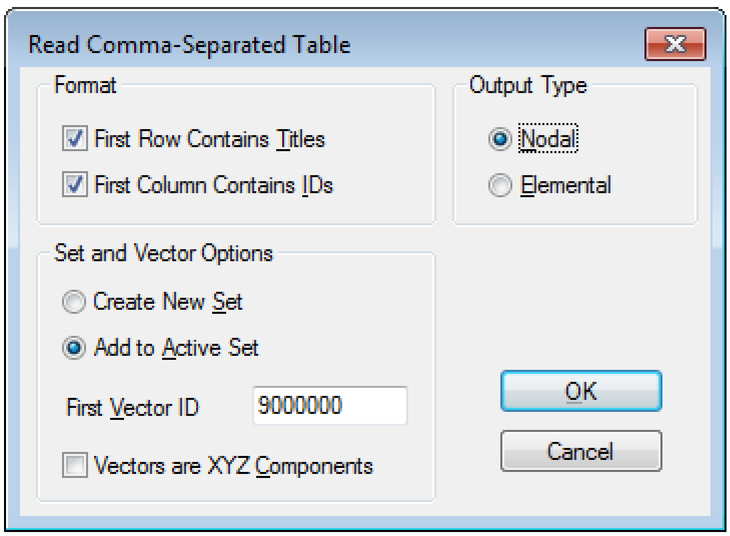
For both formats, you can also specify the number of significant digits that will be included in the output. While you can increase this number from the default, the data can only be as meaningful as the original data that you loaded into FEMAP. If you originally read output with only 6 significant digits, you can specify a larger number, but any additional digits are meaningless.

Regardless of the selected *Format*, you will then prompted to select a combination of desired Output Sets and Output Vectors to write to the CSV file using the *Select Results* dialog box after clicking *OK* in the *Write Comma-Separated Results* dialog box. For more information on using the *Select Results* dialog box, see Section 8.6.0.1, "Using the Select Output Sets and Select Results dialog boxes" of the FEMAP Commands manual.

Finally, the standard entity selection dialog box will be displayed to allow you to select the nodes and/or elements where output will be written. When using *Single Table*, you can only select nodes or elements, not both.

8.10.2 Reading or attaching to a Comma-Separated File

To read a comma-separated table, choose the *File, Import, Analysis Results* command and select the *Comma-Separated* format. You will see the standard file access dialog box, where you can choose the name of the file to read. As before, the default extension is always ".CSV". If reading a CSV file in the format described in Section 8.10.3, "The Comma-Separated Table Format", you will then see the *Read Comma-Separated Table* dialog box which is described in detail later in this section. If you are reading or attaching to a CSV file in the format described in Section 8.10.4, "The Extended Comma-Separated Table Format" (must be in this format to attach), the *Read Comma-Separated Table* dialog box will not appear, as all of the required information is specified within the file.



Read Comma-Separated Table dialog box

Format

If the first row of your file contains vector titles, you must select the *First Row Contains Titles* option. The titles will be read from the file and added to the vectors that are created. If the file does not have titles, default titles will be assigned to each new vector.

If the first column of your file contains node or element IDs, you must select the *First Column Contains IDs* option. The IDs will be read from the file and used to properly load the output data. If your file does not contain IDs, FEMAP will load the output data as if it matched the nodes/elements in your model (i.e. the first output value will be associated with the first node/element, the second with the second, and so on). You may not have any missing output values, and they must be in order.

If your file contains both IDs and titles, there must be an extra title (or simply a blank field, specified by a leading comma), for the ID column. The second title will be associated with the first output vector.

Output Type

These options specify whether the data in the file is nodal or elemental data. Since this file format is basically just a table of numbers, there is no way for FEMAP to determine what type of data you are loading. You must specify the correct option here.

Set and Vector Options

You can either read the output into a new output set, by choosing *Create New Set*, or add it to the active set, by choosing *Add to Active Set*. This second option will not be available if you do not have an active set. You must also specify the ID of the first vector to be created. Any additional vectors will be created with IDs greater than this number. The default will be the first available user-defined vector.

If the file that you are reading contains 3 vectors that are the global X, Y and Z components of some output, or 6 vectors that are the global X, Y and Z components, and the associated X, Y, Z rotations of some output, you should choose the *Vectors are XYZ Components* option. When this option is checked FEMAP will automatically calculate a 4th (or 7th and 8th) vector which is the magnitude of the component data. In addition, FEMAP will make the appropriate modifications to the vectors so that you can choose the magnitude vector and use it for displaying three-dimensional deformations and animations. If you have component data, you should always load it in this manner. Do not calculate the magnitude yourself and include it in the file. The vectors will still not be setup properly to do the deformation.

8.10.3 The Comma-Separated Table Format

The comma-separated table format that is supported by FEMAP is very simple. The following sample shows the basic capability, including titles and ID numbers, for a file with three output vectors:

```
, "title 1", "title 2", "title 3"
1, 1.0, 1.0, 1.0
2, 2.2, 3.1, 4.0
5, 1.4E2, 2.3E2, 0.111
10, , , 0.5
```

This file contains output for four entities (1, 2, 5 and 10). They could be either nodes or elements. There is no way to tell from the file. Note that since IDs are contained in the first column, the first record which contains the titles starts with a comma to skip the title for the IDs. You could also specify a title for the IDs, but it will be ignored.

The titles in the first record are text and must be less than 25 characters long each. Typically you should enclose them in quotes, as shown, but this is not required as long as no commas are contained in the title. If you specify any titles in the file, you must specify a title for all vectors.

Output data is specified in the remaining records. Each record corresponds to a node or element, and contains one output field for each vector that will be created. You can skip intermediate fields just by entering a series of commas (like entity 10 above, where the 0.5 is for the third vector). All skipped data will have the value of 0.0. If you do not specify titles in the file, the first record must contain output for all vectors. Later records can omit data for trailing output vectors if you want the values to be 0.0. The numeric values can be in floating point or exponential format. You can enclose the output values in quotes, if you want, but they are not required.

If you do not include the ID column, data for all nodes/elements must be present, and must be in numerical order corresponding to the entities in your FEMAP model. If IDs are not present, the first title will correspond to the first output vector (the leading comma should not be present).

8.10.4 The Extended Comma-Separated Table Format

The extended comma-separated table format that is supported by FEMAP is a little more complex than the original comma-separated format, but offers much more control and flexibility. The data is separated into blocks containing different “Table Types”, each of which is described in detail below. Repeat blocks in the CSV file for as many tables as you need.

Note: To attach to output using a CSV files, the files MUST use the extended comma-separated table format. See Section 2.3.2, "File, Attach to Results" in *Commands* for more information on attaching to results.

The general format of the Extend CSV Format Blocks:

Table Type
Result Type Info (Not in Table Type 100, 0=Any, 1=Displacement, 2=Velocity/Acceleration, 3=Force, 4=Stress, 5=Strain, 6=Thermal
Set/Vector ID(s) (0 for Auto)
Title(s)
... multiple rows of table data, format depends on table type ...
-1 (delimiter indicating CSV Block is finished)

Available Table Types:

100 - Output Set	100 0 (or Output Set ID) Set Title Set Value -1 NOTE: If this file is read into FEMAP, the Output Set ID is ignored - a new Output Set will be created for each Table 100 encountered. All other tables that come after this are added to that Output Set, until another Table 100 is found. All tables except this one can come in any order, or have as many occurrences as you need.
200 - Nodal Scalar(s)	200 Result Type (0 for Any) Column1 Vector ID (, Column2 Vector ID, …) (0 for Automatic IDs) Column1 Title (, Column2 Title, ….) Node ID, Value (, Value, …) … Node ID, Value (, Value, …) -1 NOTE: While multiple columns are possible in this format, for best performance when using this file as an external attached file, only 1 column should be used.
300 - Elemental Scalar(s)	300 Result Type (0 for Any) Column1 Vector ID (, Column2 Vector ID, …) (0 for Automatic IDs) Column1 Title (, Column2 Title, ….) Element ID, Value (, Value, …) … Element ID, Value (, Value, …) -1 NOTE: While multiple columns are possible in this format, for best performance when using this file as an external attached file, only 1 column should be used.
400 - Nodal Vector (Global Rectangular)	400 Result Type (0 for Any) Total Vector ID, X Vector ID, Y Vector ID, Z Vector ID (0 for Auto) Total Title, X Title, Y Title, Z Title Node ID, X Value, Y Value, Z Value … Node ID, X Value, Y Value, Z Value -1 NOTE: Although you specify a vector ID and title for the "total" values, you do not actually include those in the data. FEMAP will automatically compute the vector sum of the components and store that as the total.
401 - Nodal Vector with Rotations (Global Rectangular)	401 Result Type (0 for Any) Total ID, X ID, Y ID, Z ID, TotalR ID, XR ID, YR ID, ZR ID (0 for Auto) Total Title, X Title, Y Title, Z Title, TotalR Title, XR Title, YR Title, ZR Title Node ID, X Value, Y Value, Z Value, XR Value, YR Value, ZR Value … Node ID, X Value, Y Value, Z Value, XR Value, YR Value, ZR Value -1 NOTE: Although you specify vector IDs and titles for the "total" values, you do not actually include those in the data. FEMAP will automatically compute the vector sum of the components and store those as the totals.

500 - Elemental With Corner Data	500 Result Type (0 for Any) Centroid Vector ID, Corner1 VecID, …,CornerN VecID (0 for Auto) Centroid Title, Corner1Title, …CornerN Title Element ID, Centroid Value, Corner1 Value, …, CornerN Value … Element ID, Centroid Value, Corner1 Value, …, CornerN Value -1 ____ NOTE: Care must be taken with this format if you are writing data for Tetra or Wedge elements. For Tetra elements, corners must be 1,2,3 and 5. For wedge elements, corners must be 1,2,3,5,6,7. In both cases corner 4 is skipped. If the table contains purely Tetra or Wedge results, specify the Corner 4 VecID = -1, skip the Corner 4 Title and the Corner 4 Values with ,, … like… 500 9000000,9000001,9000002,9000003,-1,9000004 "Center Stress", "Stress C1", "Stress C2", "Stress C3",,"Stress C5" 1, 1.0, 2.0, 3.0,,5.0 …
501 - Elemental With Corner Data (not linearly combinable)	This table has the same format as 500 however, if read into FEMAP, the data will be skipped during linear combinations.
502 - Elemental Beam/Bar Data	502 Result Type (0 for Any) , OptionalRevFlag (0 for Auto) End A Vector ID, End B Vector ID End A Title, End BTitle ElementI D, End A Value, End B Value … Element ID, End A Value, End B Value -1 ____ NOTE: The OptionalRevFlag does not need to be specified. If not, the sign convention assumes that End B Values need to be reversed in sign for consistent display. If they should not be, specify OptionalRevFlag=1
503 - Elemental Beam/Bar Data (not linearly combinable)	This table has the same format as 502 however, if read into FEMAP, the data will be skipped during linear combinations.

9 Geometry Interfaces

FEMAP also contains interfaces to many CAD packages through different geometry formats. These formats include ACIS solid model file format (*.SAT File), Parasolid solid model format (*.X_T File), STEP, IGES, VDA, Stereolithography, DXF, VRML and numerous CAD native formats. Each of these formats is discussed in more detail below. Some of these interfaces are available in all versions of FEMAP; others are limited to specific versions. If you have not purchased one of the interfaces, you will be presented with a warning message to let you know that format is not available.

Geometry Format Interfaces

The following table lists the current geometry format interfaces and supported versions.

Geometry Format	Section	Related CAD Software	FEMAP Interfaces	Latest Supported Version
ACIS	Section 9.1.1, "Reading ACIS (SAT) Files"	AutoCAD	Read ACIS geometry into FEMAP and convert it to Parasolid geometry	ACIS 26 SP1
	Section 9.1.2, "Writing ACIS (SAT) Files"		Write FEMAP model as ACIS .SAT solid model file	ACIS 26 SP1
Parasolid	Section 9.2, "Parasolid Interfaces (*.X_T Format)"	Solid Edge NX Unigraphics CADKEY IronCAD Microstation MSC.Marc MSC.Patran Pro/Desktop SolidWorks	Write FEMAP model as Parasolid .X_T files Read Parasolid geometry .X_T files into FEMAP	Parasolid 29.0
	Section 9.2, "Parasolid Interfaces (*.X_T Format)"		Read Parasolid geometry into FEMAP and convert it to ACIS geometry	Parasolid 29.0
STEP	Section 9.3, "STEP Interface (*.STP files)"	CATIA I-DEAS Pro/Engineer	Write Parasolid data from FEMAP to STEP file	AP 203, AP 214 (Geometry Only)
	Section 9.3, "STEP Interface (*.STP files)"	CATIA I-DEAS Pro/Engineer	Read STEP file into FEMAP, and convert to Parasolid geometry	AP 203, AP214 (Geometry Only)
IGES	Section 9.4, "IGES File Format"	CATIA I-DEAS Pro/Engineer	Write FEMAP data as IGES files	IGES 4.0-5.3
	Section 9.4, "IGES File Format"		Read IGES data into FEMAP (types of entities that can be read vary with FEMAP configuration - see the *FEMAP User Guide* for details)	IGES 4.0-5.3
DXF	Section 9.5, "DXF Interfaces"	AutoCAD	Read DXF file into FEMAP	-

CAD Software Interfaces

The following table lists the current CAD software interfaces and supported versions.

CAD Software	Section	FEMAP Interfaces	Latest Supported Version
CATIA V4	Section 9.6, "CATIA Interface"	Read CATIA files into FEMAP	CATIA 4.1.9 to 4.2.4
CATIA V5	Section 9.6, "CATIA Interface"	Read CATIA V5 files support is available with an add-in module directly in FEMAP.	CATIA V5 R8 - V5-6 R2015 SP3
I-DEAS	Section 9.7, "I-DEAS Geometry Interface"	Write FEMAP data as IDI file for import into I-DEAS Read I-DEAS data in IDI file into FEMAP	I-DEAS 9m2
Pro/Engineer	Section 9.8, "Pro/ENGINEER Interface"	Read Pro/Engineer .PRT (single part) and .ASM (assembly) files into FEMAP	Pro/Engineer 16 - Creo 3
Solid Edge	Section 9.9, "Solid Edge Interface"	Read Solid Edge .PAR (single part), .PSM (sheet metal part), .PWD (legacy weldment), and .ASM (assembly) files into FEMAP	Solid Edge with Synchronous Technology 8
NX	Section 9.10, "NX Interface"	Read NX and Unigraphics data into FEMAP	NX 1-NX 10.0 (8.5 for 32-bit)
SolidWorks	Section 9.11, "SolidWorks Interface"	Read SolidWorks .SLDPRT (part) and .SLDASM (assembly) files into FEMAP	SolidWorks 2000 - 2016

File Formats

The following table lists the current file format interfaces and supported versions.

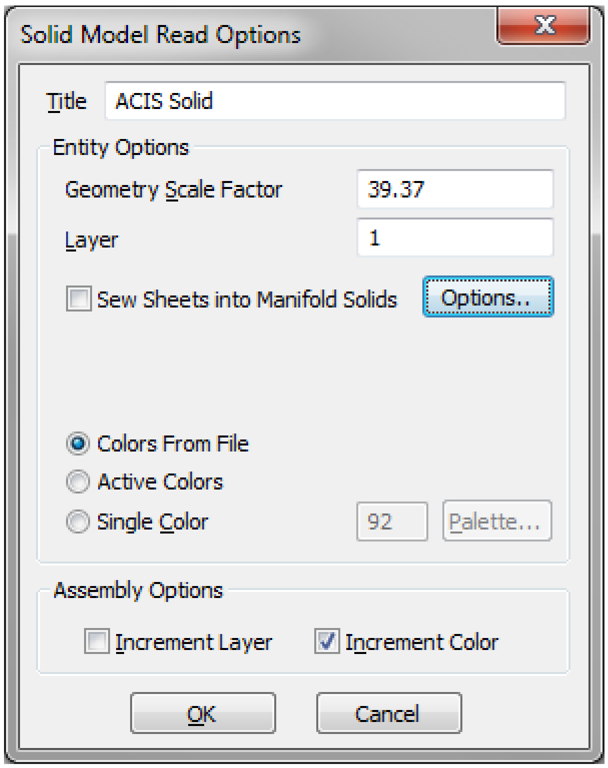
File Format	Section	FEMAP Interfaces	Latest Supported Version
Stereolithography	Section 9.12, "Stereolithography Interface"	Write FEMAP data to stereolithography (.STL) files Read stereolithography files into FEMAP	-

9.1 ACIS Interfaces (*.SAT Format)

FEMAP can import and export ACIS solid model geometry via the ACIS *.SAT file format. Many popular CAD packages such as AutoCAD are currently ACIS-based. By reading the ACIS solid model File, FEMAP provides a simple method to import solid geometry from these popular CAD packages. Since FEMAP does not incorporate the ACIS solid modeling kernel, it is necessary to convert models that are in this format to Parasolid geometry. This conversion happens automatically upon import and export and is very robust.

9.1.1 Reading ACIS (SAT) Files

Reading an ACIS File is very straightforward. Simply select the *File, Import, Geometry* command, and choose the appropriate file from the Windows *Open File* dialog box. After opening the file, you will see the *Solid Model Read Options* dialog box.



This dialog box contains three main sections: *Title*, *Entity Options*, and *Assembly Options*.

Title

FEMAP will read the *Title* of the solid model file from the SAT file, and display it. You can choose to keep this as the title of the solid, or input a new title.

Entity Options

These options affect the individual entities in the model.

Geometry Scale Factor

You can use the *Geometry Scale Factor* to change the units of the model. This can be extremely useful with some geometry packages that default to a standard for output units (such as meters) even though you may have used other units to model the part.

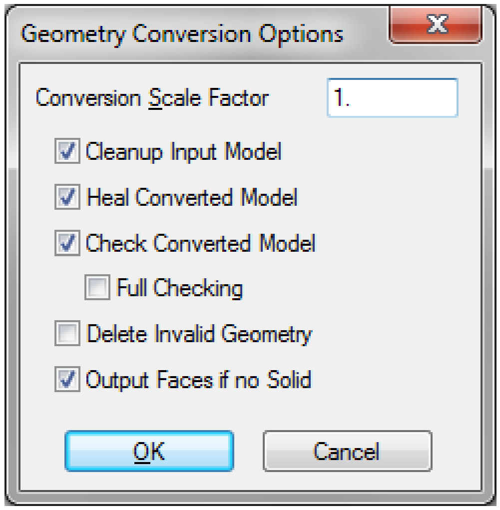
Layer

You can define the layer of the solid model.

Sew Sheets into Manifold Solids

This option controls if sheets (surfaces) should be stitched into manifold solids. If this option is on, then the surfaces will only be “sewn” together into manifold solids. It will only create “manifold solids”, as ACIS does not support “NonManifold Solids” (i.e., General Bodies) like Parasolid. If off, no “sewing” will occur.

You can also specify a number of conversion options by pressing the *Options* button, which displays the *Geometry Conversion Options* dialog box:



- The *Conversion Scale Factor* sets a scale factor that is applied to the part during conversion. Depending upon your part size, it might be necessary to modify this value to make sure that the part fits inside the Parasolid modeling box (+/- 500 units), or to achieve a consistent size, with other geometry. Unlike the geometry scale factor that is normally applied, this factor actually changes the dimensions of the underlying geometry (the other is simply a factor used for presenting the dimensions to you).
- *Cleanup Input Model* can be used to detect and attempt to cleanup sliver faces, or other problem geometry during the conversion.
- *Heal Converted Model* attempts to close gaps and correct geometric inaccuracies if it is turned on.
- *Check Converted Model* will run Parasolid geometry checking on the converted model to attempt to find and report any remaining problems. If you also turn on *Full Checking*, checks are also made for any surface discontinuities or irregularities. These checks take longer and may not be necessary, so this option is off by default.
- *Delete Invalid Geometry* will delete the converted geometry if it does not pass the checking options.
- *Output Faces if no Solid* will produce unstitched, free faces even if it is not possible to do a full conversion of the original solid. This allows you to get partial conversion.

Color

You can choose to use the Colors From File, Active Colors, or Single Color. When Colors From File is used, FEMAP will use color data found in the geometry file and match it as closely as possible to a color in FEMAP. Active Colors will use the default colors in FEMAP for geometry, while Single Color allows you to choose a color from the *Palette*.

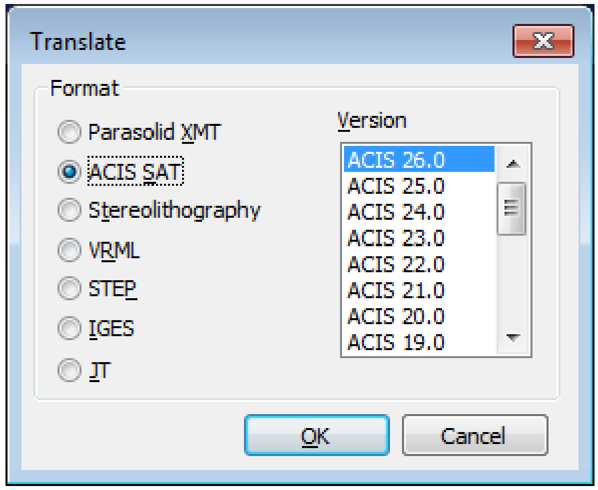
Assembly Options

FEMAP also has the capability to read ACIS Assembly files and converts them to Parasolid as well. You can choose to have each solid from the assembly on a different layer, as well as a different color.

Once you select *OK*, FEMAP will open the ACIS-to-Parasolid converter and read the file. No other action is required.

9.1.2 Writing ACIS (SAT) Files

FEMAP can also export an ACIS solid model file. Select the *File, Export, Geometry* command, and you will see the *Translate* dialog box:



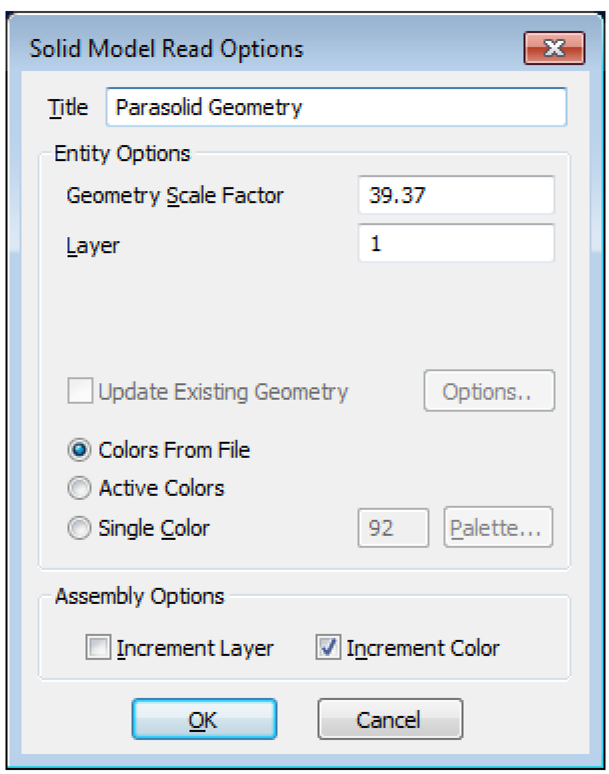
Choose ACIS SAT file and the appropriate version (ACIS 7.0 and above). You should always use the latest version of ACIS unless you plan to import the file to a CAD package that only supports an earlier version of ACIS. You will then be prompted for the solids to export and the file name.

9.2 Parasolid Interfaces (*.X_T Format)

FEMAP can import and export Parasolid solid model geometry via the Parasolid transmit file (*.X_T) format. Many popular CAD packages such as Solid Edge, SolidWorks and Unigraphics are Parasolid-based. By reading the Parasolid solid model file, FEMAP provides a simple method to import solid geometry from these popular CAD packages. Since FEMAP also incorporates the Parasolid solid modeling kernel, they are read by the same modeling software that originally created them. This makes reading and writing X_T files very robust.

9.2.1 Reading Parasolid (X_T) Files

Reading a Parasolid file is identical to reading an ACIS file, except you must choose the *Parasolid Transmit (X_T)* file. For information on the options available, see Section 9.1.1, "Reading ACIS (SAT) Files".



The *Update Existing Geometry* option is described below.

Update Existing Geometry

This option lets you import an updated version of the Parasolid model. For example, if you changed a dimension, FEMAP will automatically update the following entities to match the new geometry:

- colors and layers
- settings for *Mesh, Mesh Control, Approach on Surface*
- settings for *Mesh, Mesh Control, Attributes*
- settings for *Mesh, Mesh Control, Mesh Points on Surface*
- settings for *Mesh, Mesh Control, Custom Size Along Curve*
- geometric constraints and loads
- connections (surface-based Connection Regions and Connectors)
- boundary surfaces that use updated surfaces or curves
- groups

The mesh itself will not automatically be updated.

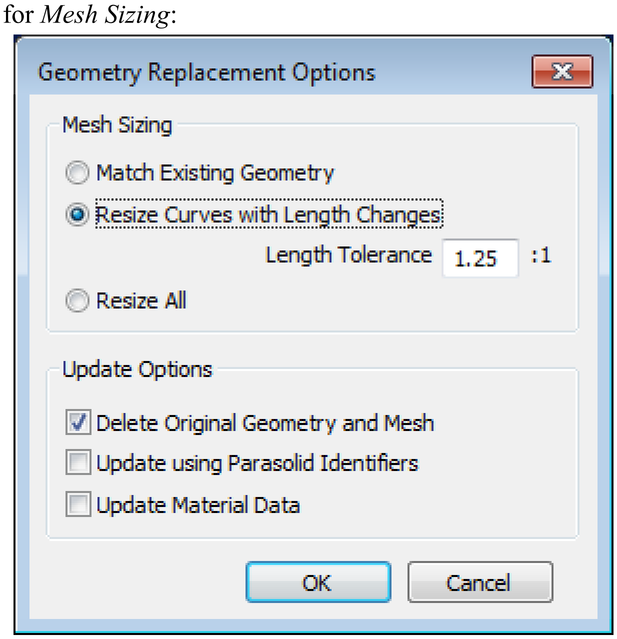
Note: FEMAP uses Point, Curve, and Surface IDs to retain FEMAP specific information (mesh sizing, approaches on surfaces, colors, geometric based loads and boundary conditions, etc.) when a modified solid is imported into FEMAP using the *Update Active Solid* option. Most topological changes to the geometry (fillets, chamfers, thru holes, bosses, notches, etc.) will likely alter some the IDs in the geometry. Once these IDs have been changed, FEMAP is NOT able to “map” the information onto any entity with an altered ID. Be sure to thoroughly examine the model before analysis in order to assure accuracy. In some cases, the information on a certain entity will be completely lost and must be re-applied.

For example, an existing model has a geometry-based load on surface 10 in FEMAP. In another Parasolid-based CAD tool, a fillet is placed along the edge of the loaded surface and another surface. The fillet changes the overall shape and size of the surface and alters the ID to 20. When the modified geometry is imported into FEMAP, surface 10 does not exist in the new geometry and FEMAP will not be able to re-apply the geometry-based load to surface 10 and will not make any attempt to “map” the load which was on surface 10 to surface 20.

Geometry Replacement Options

Pick the *Options* button to set options on the *Geometry Replacement Options* dialog box. These include *Mesh Sizing* and *Update Options*.

There are three options for *Mesh Sizing*:



- *Match Existing Geometry:* keep the mesh sizing the same as that on the original geometry. If there are eight elements on an original curve, there will be eight elements on the updated curve.
- *Resize Curves with Length Changes:* resizing will occur when the ratio between new and old geometry exceeds the tolerance that you enter. The value that you enter is always greater than 1.0, but it applies to curves that lengthen or shorten.
- *Resize All*

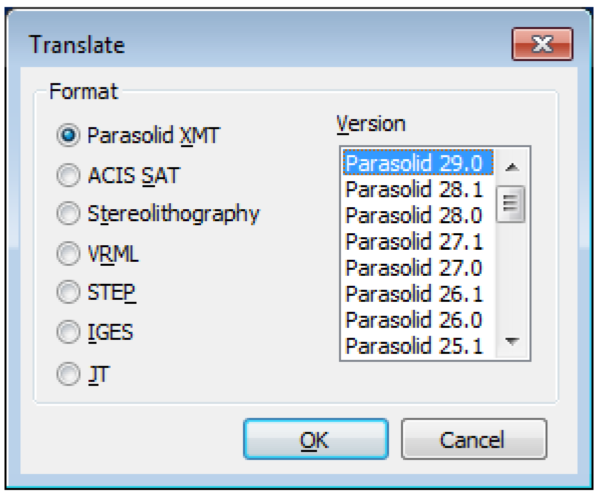
Under *Update Options,* by default the software will delete the original geometry and mesh.

You can also choose to have FEMAP *Update using Parasolid Identifiers* or use the “Solid Edge” method. The “Solid Edge” method is on by default and will improve efficiency and accuracy when bringing in Parasolid geometry from Solid Edge and should have no effect on Parasolid geometry coming from any other CAD package.

If there is an issue using the Solid Edge method, turn *Update using Parasolid Identifiers* option on. *Update Material Data* will change the material which was automatically created in FEMAP when the model was originally read in from Solid Edge.

9.2.2 *Writing Parasolid (X_T) Files*

FEMAP can also export a Parasolid solid model file via the X_T (Transmit) format. Simply select the *File, Export, Geometry* command, select *Parasolid XMT* file, and the appropriate version. You should always use the latest version of Parasolid unless you plan to import the file to a CAD package which only supports an earlier version. You will then be prompted for the solids to export and the file name.



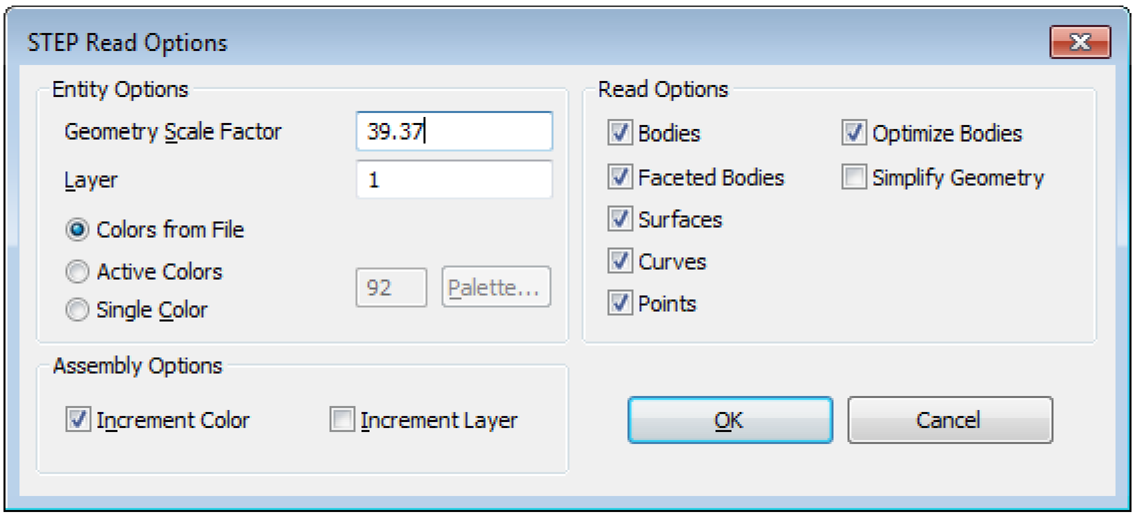
9.3 STEP Interface (*.STP files)

STEP, the Standard for the Exchange of Product Model Data, is a comprehensive ISO standard (ISO 10303) that describes representation and exchange of product information. Product data must contain enough information to cover a product's entire life cycle, from design to analysis, manufacture, quality control testing, inspection and product support functions. In order to do this, STEP covers geometry, topology, tolerances, relationships, attributes, assemblies, configuration and more.

The FEMAP STEP interfaces currently focus on geometry and topology transfer. They can import and export STEP AP203 Solid Entities and STEP AP214 Class II, III, IV, V, and VI entities. This provides a convenient method of interfacing with CAD packages that are not ACIS or Parasolid based.

9.3.1 Reading STEP (*.STP) Files

Reading a STEP file is procedurally identical to reading an ACIS or Parasolid file, except you must choose a STEP (*.STP) file. For more information on the options available, see Section 9.1.1, "Reading ACIS (SAT) Files". FEMAP automatically creates Parasolid geometry from the data in the STEP file.



Read Options are available to control how the translations of specific STEP entities are controlled.

Options include (all options on by default unless otherwise noted):

- *Bodies* - when on, imports manifold solid B-Rep entities
- *Faceted Bodies* - when on, imports faceted B-Rep solid entities (i.e.,often found in a JT file)
- *Surfaces* - when on, imports shell-based surfaces and surfaces without topology
- *Curves* - when on, imports wireframe curves
- *Points* - when on, imports wireframe points
- *Optimize Bodies* - when on, attempts to heal edges, remove redundant topology, and share geometry
- *Simplify Geometry* - when on, attempts to recover the analytic definitions for the B-spline geometry in the part (Off by default)

9.3.2 Writing STEP (*.STP) Files

FEMAP can also export a STEP file. The STEP file translator converts a Parasolid solid into a STEP format. Select the *File, Export, Geometry* command, select *STEP* in the *Format* section, then select the version of STEP to use from AP204, AP203 Edition 2, or AP214. You will then be prompted for the solids to export and the file name.

Note: The STEP option, which converts between a Parasolid solid to a STEP format, is only available when *Geometry Engine* is set to “1..Parasolid” on the *Geometry/Model* tab in the *Preferences* dialog box.

9.4 IGES File Format

IGES stands for Initial Graphics Exchange Specifications. IGES is a neutral file format defined by ANSI for exchange of CAD data across heterogeneous systems. IGES has been a very popular way of exchanging data and many systems provide interfaces to exchange information through this format. The IGES file supports representation of 3D geometry and topology information. It also allows representation of drawing information, symbols views etc. IGES files do not support representation of assembly and features information of models.

The IGES standard version 4.0 does not support representation of solids. So solids are represented as trimmed surfaces in IGES files produced having compliance to standard 4.0. But, later IGES standards (version 5.2 onwards) are enhanced to represent solid information (called MSBO-Manifold Solid B-Rep Object).

IGES Files with no “Start Section” may be imported as well.

9.4.1 Reading IGES Files...

FEMAP actually contains several levels of IGES read interfaces with capabilities that are appropriate to the other geometry capabilities of the version of FEMAP that you are using.

	FEMAP IGES Readers	
IGES Entity Type	**Standard Interface**	**Alternate Interface**
100 - Circular Arc	X	X
102 - Composite Curve	X	X
108 - Bounded Plane	X	X
110 - Line	X	X
112 - Parametric Spline Curve	X	X
116 - Point	X	X
118 - Ruled Surface	X	X
120 - Surface of Revolution	X	X
122 - Tabulated Cylinder	X	X
124 - Transformation	X	X
126 - Rational B-Spline Curve	X	X
128 - Rational B-Spline Surface	X	X
104 - Conic Arc	X	X
142 - Curve on Parametric Surface	X	X
144 - Trimmed Surface	X	X
106 - Copious Data	X	
114 - Parametric Spline Surface	X	
123 - Direction	X	
130 - Offset Curve	X	
140 - Offset Surface	X	
141 - Bounded Entity	X	
143 - Bounded Surface	X	
186 - MSBO (Solid)	X	
190 - Plane Surface	X	
192 - Right Circular Cylinder	X	
194 - Right Circular Conical Surface	X	
196 - Spherical Surface	X	
198 - Toroidal Surface	X	
308 - Subfigure Definition Entity	X	
402 - Associative Instance Entity	X	
408 - Subfigure Instance Entity	X	
502 - Vertex List	X	
504 - Edge List	X	
508 - Loop	X	
510 - Face	X	
514 - Shell	X	

Although the Parasolid option is the preferred method of importing complex solid and surface geometry into FEMAP, the FEMAP IGES interfaces provide a robust capability to import complex geometry from non-Parasolid based CAD packages.

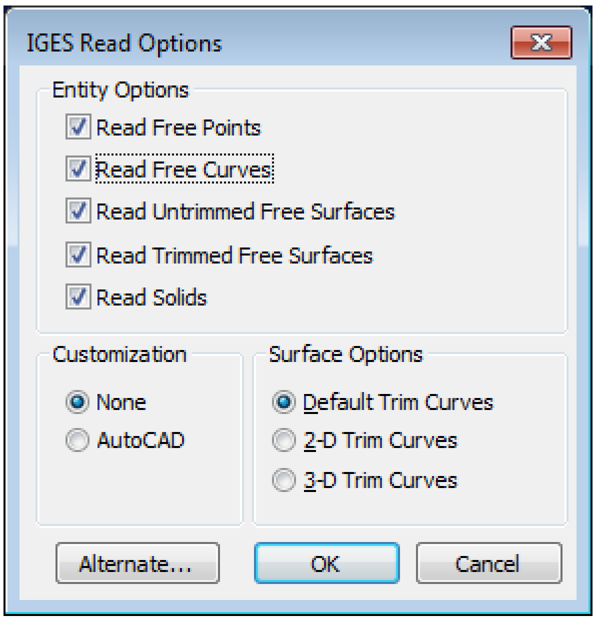
FEMAP can read simple IGES curves and surfaces, as well as trimmed surface or solid data. In general, when working with solid models from CAD systems, or complex surfaces models, one of these systems will be required. In these cases, the geometry will be loaded into the Parasolid geometry engine.

Standard IGES Interface

To read an IGES file, select the *File, Import, Geometry* command and choose the appropriate file. You will then see the *IGES Read Options* dialog box. The Standard IGES translator will read in all of the entity types listed in the table at the beginning of this section.

When you are using the IGES translator, you will see the *Alternate* button at the bottom of the dialog box. If you press *Alternate*, you will only have access to the entities at the Alternate interface level listed above. Note that these are two distinct interfaces - if you are having trouble with one of them, it can still be worthwhile trying the other.

The *IGES Read Options* dialog box is partitioned into three major sections:



Entity Options

The options available to you in the IGES interface include the ability to read or skip free points, free curves, trimmed or untrimmed free surfaces and solids. If you are having trouble reading solid data, you should try turning off solids, and reading just the trimmed surfaces.

Surface Options

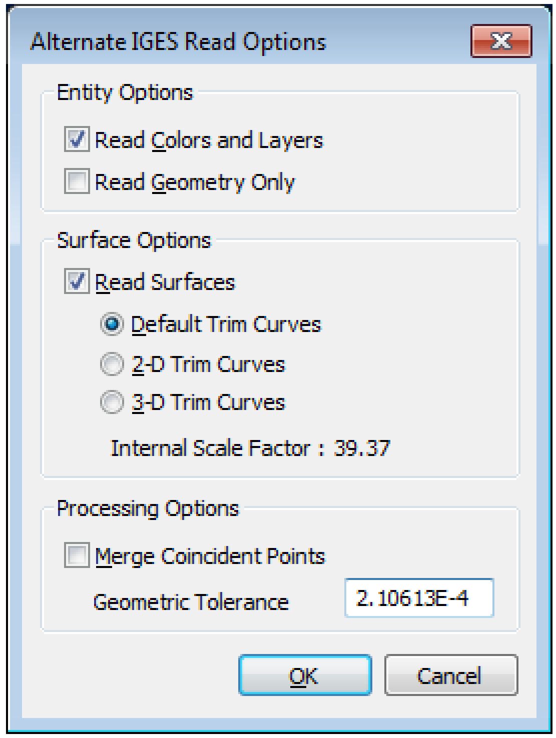
You have the ability here to control which trimming curves to read - default, 2D or 3D. In some cases, taking the default will not work, because the curves written to the file will be incorrect, even though the writing system specified them as the default. In this case, explicitly picking the other type of trimming curves will probably work better.

Customization

Finally, if you are trying to read an IGES file from some older versions of AutoCAD, you should specify the AutoCAD customization. In particular, this option corrects several problems that exist in solids written by AutoCAD Release 13.

Alternate IGES Interface

The Alternate IGES Read Options dialog box is also broken into three sections:



Entity Options

These options allow you to read (or omit) colors and layers from the IGES file. If you do not read the color and layer information from the IGES file, FEMAP will assign default values to each entity.

You may also select the *Read Geometry Only* option to simplify the model. This can be a convenient method of removing much of the construction lines and other non-geometry information that may interfere with the viewing of the model.

Surface Options

These options provide control over reading of surfaces. In general you will want to read surfaces and choose *Default Trim Curves*. If FEMAP is encountering problems with the trimming, you may want to choose *2-D Trim Curves* or *3-D Trim Curves*. Each trimmed surface in the IGES file has both 2-D (curves in surface space) and 3-D (curves in 3-D space) trimming curves, as well as a suggestion on which curve to use. If you select *Default Trim Curves,* FEMAP will use those suggested by the IGES file. The other options will use all 2-D Surface or 3-D space curves for trimming. The *Trim Curves* section is grayed if you choose not to read surfaces.

The *Internal Scale Factor* is used to reduce the size of the part in the FEMAP database. The internal engine of Parasolid requires all positions be in a box of +/- 500. If you have entities outside of this box, Parasolid cannot perform operations on them. By using an internal scale factor, you can have FEMAP scale the part internally to prevent the part from extending beyond this box. You will not see changes in the dimensions of the part since FEMAP will do all scaling internally. This option allows the input of very large dimensions for the model, without exceeding the limits of the Parasolid geometry engine.

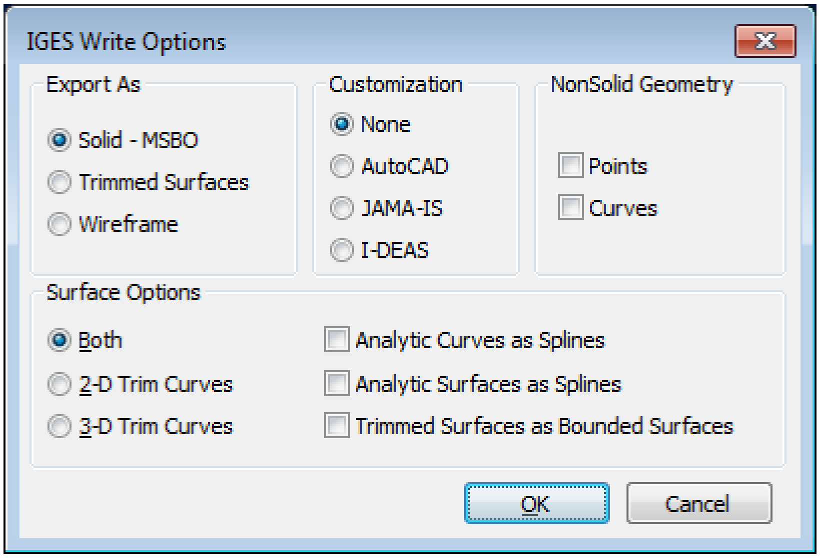
Hint: If FEMAP cannot trim some of your surfaces, it may be advantageous to read the IGES file a second time, selecting *Read Geometry Only* and deselecting *Read Surfaces*. FEMAP will import the original curves, and you can then use these curves in FEMAP to modify (or trim) the surfaces which were not trimmed in the importing process.

Processing Options

When *Merge Coincident Points* is on, FEMAP will eliminate points that are within the *Geometric Tolerance* from each other after the model has been translated. If you leave this option off, you can still merge the points later with the *Tools, Check, Coincident Points* command. The default minimum distance is loaded from the IGES file that you are reading, but you may modify this value.

9.4.2 Writing IGES Files...

When you are using the IGES translator, you will see an IGES option when you choose *File, Export, Geometry.* When you select that option, you will also see an *Options* button. Normally, to write an IGES file, you will not need to change any of these options unless you want to customize how the data will be written.



Only Parasolid geometry can be written using the IGES interface. You can export the geometry as solids, trimmed surfaces or wireframe data. The option that you choose will depend on the system where you will be reading the IGES file. Options to export non-solid geometry such as free points and/or curves are also available.

If your system does not support solid data, switch the *Export As* option to trimmed surfaces for best results. Similarly for trimming curves, if you want to use either 2-D or 3-D curves exclusively, pick one of those surface options - otherwise both will be written. If you want to write all curves or surfaces as splines (not lines, arcs, circles...) then check one or both of those surface options. Finally, the *Trimmed Surfaces as Bounded Surfaces* option controls which surface entities are created. If this option is off, surfaces will be converted to Type 144 - trimmed surfaces. If it is on, they will be converted to Type 143 - bounded surfaces.

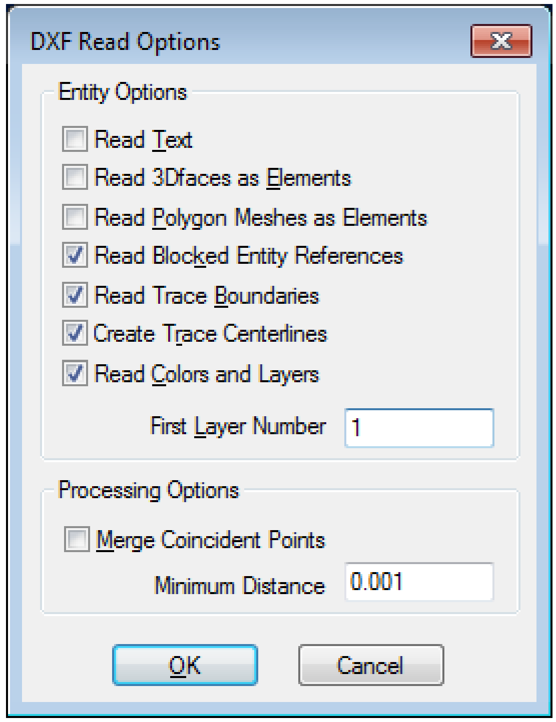
If you are trying to send data to a particular CAD system, or using a particular standard, you may want to choose one of the *Customization* methods for better results. These flavors will tend to override other settings that you make for the *Surface Options* and *Export As Options*.

9.5 DXF Interfaces

FEMAP can also import geometry (points, curves...) from most popular CAD systems via a DXF formatted file. Although the DXF format originated in AutoCAD, most CAD systems can export data in this format. FEMAP cannot currently export a DXF file.

Reading the File

Choose the *File, Import, Geometry* command and select the desired DXF file. You will then see the *DXF Read Options* dialog box:



The entity options allow you to selectively read, or skip, certain entities that are in the DXF file.

Read Text

If you turn this option on, any text or notes that are in your DXF file will be read and converted to text in your FEMAP model.

Read 3Dfaces as Elements

3Dfaces are used to represent triangular or quadrilateral polygons. If you turn this option on, FEMAP will read the faces and convert them to plate elements. If the option is off, FEMAP simply creates the lines that define the edges of the face.

Read Polygon Meshes as Elements

This is similar to the previous option except it applies to polygon/polyface meshes that are in the file.

Read Blocked Entity References

If your DXF file contains blocked entities, FEMAP will only read them if this option is on. Normally, you will want to read the entire model so leave this option on. If you want to skip blocked entities however, you can turn this off. You can then control what will be read by selectively exploding block references before you create the DXF file.

FEMAP cannot read arrays of inserted blocks. In this case you will have to explode at least those references. Similarly, blocks that are inserted with different scale factors in different directions cannot be read. If possible, you can try exploding them (AutoCAD cannot explode this type of block reference).

Read Trace Boundaries

Traces are lines that have a width defined, and therefore are represented by a rectangular boundary. If you turn this option on, four lines are created to represent the edges of the trace. If you turn it off, you can still get the centerline of the trace with the next option.

Create Trace Centerlines

If you turn this option on, lines are created to represent the centerlines of all traces.

Read Colors and Layers

If you want FEMAP to assign colors and layers to the geometry and text that it reads based on the information in the DXF file, turn this option on. When it is off, colors and layers will be assigned using FEMAP's defaults. Since DXF layers can have any name, and FEMAP layers are purely numbers, FEMAP will create a unique number for each unique layer name. The first layer name will be assigned to the layer number that you choose in the *First Layer Number* option.

Merge Coincident Points

When FEMAP reads geometry from your file, each curve will have unique endpoints, even if they are coincident with the end of another curve. To use the geometry in FEMAP, you will probably want to join curves that have coincident ends to the same point. If you turn on this option, FEMAP will do this for you. In addition, you can define the *Maximum Distance* that points can be apart and still be considered as coincident.

Supported DXF Entities

The following entities are supported by the DXF Read Translator:

Entity	Group Codes	Description
POINT	6	Linetype
	8	Layer
	10,20,30	Point
	38	Elevation
	62	Color
	210,220,230	Extrusion Direction
LINE	6	Linetype
	8	Layer
	10,20,30	Start Point
	11,21,31	End Point
	38	Thickness
	62	Color
	210,220,230	Extrusion Direction
	Lines are always translated to FEMAP lines. If a thickness is specified, a single DXF line creates 4 FEMAP lines which represent the original “thick” line.	
CIRCLE	6	Linetype
	8	Layer
	10,20,30	Center Point
	40	Radius
	38	Elevation
	62	Color
	210,220,230	Extrusion Direction
	Circles translate to FEMAP circles. Additional points are created as required to define the FEMAP circle.	
ARC	6	Linetype
	8	Layer
	10,20,30	Center Point
	40	Radius
	50	Start Angle
	51	End Angle
	38	Elevation
	62	Color
	210,220,230	Extrusion Direction
	Arcs translate to FEMAP arcs. Additional points are created as required to define the FEMAP arc.	

Entity	Group Codes	Description
TRACE	6	Linetype
	8	Layer
	10,20,30 11,21,31 12,22,32 13,23,33	Corner Points
	38	Elevation
	62	Color
	210,220,230	Extrusion Direction
	Traces are read as FEMAP lines. Options are available to create the outline and/or the centerline of the trace, as specified by the four corners.	
TEXT	1	Text Value
	6	Linetype
	8	Layer
	10,20,30	Insertion Points
	38	Elevation
	62	Color
	72,73	Horizontal and Vertical Justification
	210,220,230	Extrusion Direction
	Text can be read as FEMAP text entities. It is usually best to skip text however, since font and orientation information cannot be translated.	
SOLID	6	Linetype
	8	Layer
	10,20,30 11,21,31 12,22,32 13,23,33	Corner Points
	38	Elevation
	62	Color
	210,220,230	Extrusion Direction
	Solids are read just like 3DFACEs.	
3DFACE	6	Linetype
	8	Layer
	10,20,30 11,21,31 12,22,32 13,23,33	Corner Points
	38	Elevation
	62	Color
	210,220,230	Extrusion Direction
	3DFACES can either be translated to lines which define the outline of the face, or directly to Plate elements.	
POLYLINE	6	Linetype
	8	Layer
	38	Elevation
	62	Color
	70	Polyline Flag
	71, 72	Polygon Mesh M and N Vertex Counts
	73, 74	Smooth Surface M and N Densities
	75	Smooth Surface Type
	210,220,230	Extrusion Direction

Entity	Group Codes	Description
POLYLINE VERTICES	8	Layer
	10,20,30	Point
	38	Elevation
	62	Color
	70	Vertex Flag
	71,72,73,74	Vertex Indices for Polyface Mesh
	DXF Polylines represent a variety of curve and surface types. FEMAP converts all polylines which represent curves to a series of line segments (either lines or plot-only elements). FEMAP converts polygon and polyface meshes into lines or plate elements (depending on your selection for the *Read Polygon Meshes as Elements* option). FEMAP does not create spline curves, or surfaces from polylines.	

In addition to the above entities, the LAYER table is read if it is present. This enables FEMAP to translate from the DXF layer names into FEMAP layer numbers and supports reading the Group Code 8 blocks defined above.

Exploding DXF Blocked Entities

Other than the LAYER table, FEMAP only reads the ENTITIES section of the DXF file. The BLOCKS section is skipped. If you have blocked data, you should use the EXPLODE command prior to writing the DXF file for FEMAP. If you read a DXF file, and portions of the geometry are missing, go back to AutoCAD, EXPLODE that portion of your model, and write a new DXF file. You may have to use EXPLODE several times if you have nested BLOCKS (if you have a solid model in AutoCAD you will have to use EXPLODE at least twice to obtain the points and curves). You can also use this process in reverse. If there are portions of your model that you want to ignore as you translate to FEMAP, simply BLOCK them before you write the DXF file.

DXF Paper Space and Viewports

FEMAP does not support AutoCAD Paper Space, but will read all entities as if they were defined in model space. If you have “Paper Space” geometry, it will probably be positioned incorrectly relative to your model. In general, Paper Space geometry should not be included in the DXF file since it is usually not part of the defining model geometry. Likewise, FEMAP ignores Viewport information since it is trying to transfer the model rather than the correct “drawing”.

9.6 CATIA Interface

A CATIA model typically contains following categories of data:

- geometry elements (SPACE / DRAW)
- standard / user-defined attributes
- parts/assemblies
- surface and solid history information
- solid feature information
- header information (user comments, dates, name, purpose, etc.)
- model information (version number, unit, scale, tolerance, etc.)

A model file is the native CATIA file format containing basic geometry. It may point to a project file for accessing some table related to annotation styles, etc. (like the hatch pattern style). The file extension is *.model. The *.model file may contain either single or multiple solids, with or without a combination of space and/or draw entities.

About CATIA Export Files

An export file is a combination of all model file data and all necessary information from project files referenced by model files. The extension is typically .exp. It can contain single or multiple models and project information (CATIA setups). The files are created with the CATIA CATEXP utility.

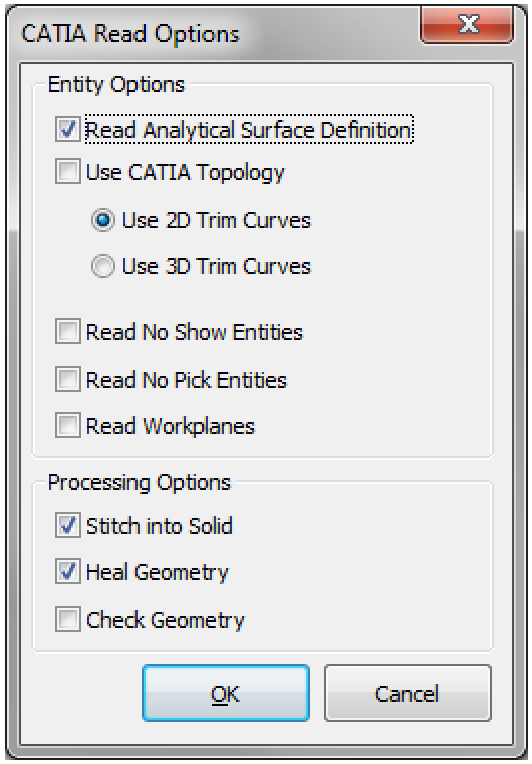
CATIA Entity	Supported
SPACE elements	
Point	Yes

CATIA Entity	Supported
Line	Yes
Circle	Yes
Ellipse	Yes
Parabola	Yes
Hyperbola	Yes
Polynomial Curve	Yes
Bspline Polynomial Curve	Yes
NURBS Curve	Yes
Composite Curve	Yes
Cloud of points	No
Plane	Yes
Polynomial Surface	Yes
Bspline Polynomial Surface	Yes
Polyhedral Surface	No
NURBS Surface	Yes
Edge	Yes
Face	Yes
Volume	Yes
Polyhedral Solid	No
Exact Solid	Yes
Transformation	Yes
Skin	Yes
Net	Yes
Ditto	No
MODEL Information	
Unit	Yes
Scale	Yes
Tolerance	Yes
HEADER Information	
Version	Yes
User	Yes
Date	Yes

9.6.1 Reading CATIA V4 Files...

To import a CATIA V4 file, choose *File, Import, Geometry* and choose the appropriate filename. If your file has a different filename extension than the defaults (.MDL, .MODEL, .EXP or .DLV), drop down the *File Type* list and choose *CATIA V4* (rather than *All Geometry*), then specify the complete filename - this will allow FEMAP to recognize that you are reading a CATIA model.

Use the *CATIA Read Options* dialog box to define options for reading in the CATIA file.



Read Analytical Surface Definition

Set this option to use analytical surface definition for surfaces represented as splines.

Use CATIA Topology

Pick this option to read the CATIA topology. Generally, it is better to leave this option off and read the surfaces in as individual surfaces. You can then use the *Stitch into Solid* option to create the solid.

Use 2D Trim Curves/Use 3D Trim Curves

Use these options to specify whether to use 2D (parametric) or 3D trim curves.

Read No Show Entities/Read No Pick Entities/Read Workplanes

These options may be used in any combination to import “No Show Entities”, “No Pick Entities”, and /or “Workplanes” (as a surface).

Stitch into Solid

Turn on this option to have the Parasolid conversion stitch surfaces together as a solid. If this option is off, the model will be transferred as a collection of surfaces which can later be stitched in FEMAP.

Heal Geometry

Turn on this option to let the interface repair tolerance and gap problems.

Running the Interface

If you open the CATIA Import window while the interface is running, you will often see many messages being displayed. You should not be alarmed by this, it does not necessarily indicate that there are any problems that cannot be converted.

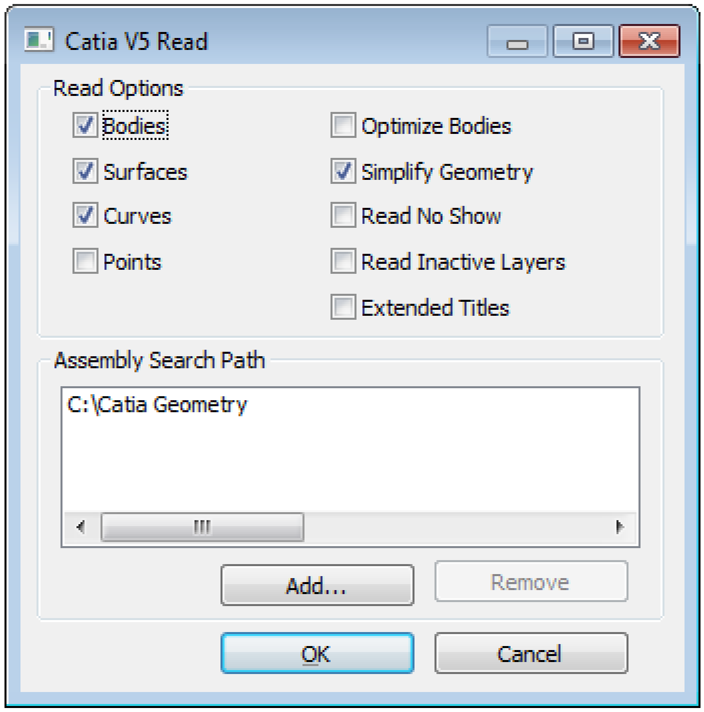
For example, you may see the message “discontinuous 2D curve”. This indicates that the interface has found some discontinuity in the CATIA definition that cannot be represented in Parasolid. In many cases, if you see this message you will get better results if you convert again using 3D trim curves.

Likewise, you might see messages that say “PK_ERROR_mild”. These normally indicate that some function in the Parasolid conversion is finding some problems with the geometry, but the geometry can usually be recovered. If you see, “surface geom check failed”, “surface not created”, “PK_ERROR_serious”, “PK_ERROR_fatal” or a similar message, it indicates that a more significant error has occurred on one or more surfaces and that the geometry may not be completely translated. Even if you do get one or more of these messages, the interface will still attempt to translate as much geometry as it can.

9.6.2 Reading in CATIA V5 Geometry

To import a CATIA V5 file, choose *File, Import, Geometry* and choose a file with a .CATP (CATIA Part) extension.

Use the *CATIA V5 Read* dialog box to define options for reading in the CATIA V5 file.



Read Options

There are several options available to you in order to get only the portions of the CATIA part that you need read into FEMAP. *Points*, *Curves*, *Surfaces*, and *Bodies* are simple on/off toggles for those geometric entity types.

- *Optimize Bodies* - when on, attempts to heal edges, remove redundant topology, and share geometry
- *Simplify Geometry* - when on, attempts to recover the analytic definitions for the B-spline geometry in the part (Off by default)
- *Read No Show* - when on, No Show entities will be converted to Parasolid (Off by default)
- *Read Inactive Layers* - when on, entities on Inactive Layers will be converted to Parasolid (Off by default)
- *Extended Titles* - when on, the “titles” of parts which have a “title” in the geometry file(s) will be appended to include Part Number (including Embedded Revision), Nomenclature, and Description information (Off by default). For any part which does not have a “title” in the geometry file(s), a title will be created using Part Number (including Embedded Revision), Nomenclature, and Description information, regardless of the setting for *Extended Titles*.

Assembly Search Path

This functionality allows you to read in an assembly file from CATIA V5 and designate search paths to individual parts located in different directories. This allows you to have the individual parts associated to an assembly file located in any number of directories.

You can use the *Add...* button to specify different directory locations or choose a listed path and remove it with the *Remove* button.

Note: Special note about the CATIA V5 translator - This product includes software developed by the Apache Software Foundation (http://www.apache.org/).

9.7 I-DEAS Geometry Interface

FEMAP can import I-DEAS Master Modeler geometry via the Interoperability Data Interface (IDI) file format that was added in I-DEAS 8. By reading the IDI file, FEMAP provides a simple method to import solid geometry from I-DEAS. FEMAP can read I-DEAS parts; assemblies are not currently supported.

Note: If you are using a version of I-DEAS later then V9 release 2, you should write a Parasolid transmit file, *.X_T, out of I-DEAS instead of an *.idi file. I-DEAS now uses the Parasolid geometry kernal, so the transfer will be direct and more complete than using the *.idi file.

Writing an IDI file from I-DEAS 8

To write I-DEAS geometry to an IDI file, follow these steps:

1. Before starting I-DEAS, add the following entry to your I-DEAS parameter file:

```
MM.Export.Ca.InMenuSw: 1
```

2. Start I-DEAS and get part onto the workbench.

3. Pick *File, Export*.

4. Select the IDI file format.

5. Pick the part to be exported.

Note: Although you can also pick assemblies, FEMAP does not support assemblies in IDI files.

6. On the *I-DEAS to 'IDI' Translator* form, enter a file name and then pick *Export.*

7. Take the defaults for the first two menus.

8. On the third menu, set *Precise B-Rep* to *On*.

9. Take defaults for all remaining prompts.

Writing an IDI file from I-DEAS 9

To write I-DEAS geometry to an IDI file, follow these steps:

1. Start I-DEAS and get part onto the workbench.

2. Pick *File, Export*.

3. Select the Viewer XML (IDI) file format.

4. Pick the part to be exported.

Note: Although you can also pick assemblies, FEMAP does not support assemblies in IDI files.

5. On the *I-DEAS to 'IDI' Translator* form, enter a file name.

6. Open the Options form and ensure *Precise Geometry Tessellator* is set to *External* and *Keep precise* is *On*.

7. Open the *Advanced* options form and ensure *Parts* is switched *On*.

8. Open the *Parts* form and ensure *Precise B-Rep* is *On*.

9. *OK* out of the options forms and then select *Export*.

Reading an IDI file into FEMAP

Reading an IDI file is very straightforward. Select the *File, Import, Geometry* command, choose the appropriate file from the Windows *Open File* dialog box, and the file will be read. The geometry will be unstitched - use the *Geometry, Solid, Stitch* command to create solid geometry.

The geometry created in FEMAP will use the same units that were active in I-DEAS when the IDI file was written.

In FEMAP, the default IDI reader is I-DEAS 9m1. This reader should read 9m1 and all previous versions. An I-DEAS 8 IDI reader and a version of I-DEAS 9m2 are also included.

9.8 Pro/ENGINEER Interface

Pro/E model data is stored in .prt files. There are two types of .prt files; uncompressed and compressed files.

The FEMAP Pro/E translator supports reading of solid, surface and wire-frame entities into FEMAP Parasolid geometry. The interface is completely independent of Pro/E - you do not need a copy of, or a license for Pro/E on your system to be able to read the part files.

Pro/E .prt files can be output in compressed or uncompressed formats. Currently only uncompressed files can be supported by this interface. If you normally save your files in compressed format, you will have to change your settings to output an uncompressed file for conversion to FEMAP.

The interface supports all geometric and topological entities available in the Pro/E files. It does not include support for Assembly and Parametric information present in the Pro/E file. Similarly the attributes are system specific and hence they will not be supported.

Assembly files (.asm files) can be read into FEMAP. Each part of the assembly will come in as a separate solid in FEMAP.

Note: In order for an assembly (.asm) file to be read into FEMAP correctly, all of the part (.prt) files which are referenced in the assembly file MUST be in the same directory as the assembly file.

9.8.1 Reading Pro/E Files...

To import a Pro/E file, select *File, Import, Geometry* and choose the appropriate filename. Since Unigraphics also uses the .PRT extension, when you have the File Type set to *All Geometry,* FEMAP attempts to automatically distinguish and choose the right interface to read the file that you select. If it cannot determine the type automatically, it will ask you to choose which interface you want. If you are having problems with this, open the *File Type* list and directly choose the Pro/E interface.

There are no additional options to specify for this interface. It simply reads the Pro/E file and loads it into FEMAP.

Note: If you are having trouble importing a model with the direct Pro/E interface, or if it is a problem to produce an uncompressed model file, try using the IGES interface. The IGES files written by Pro/E work very well with the IGES interfaces in FEMAP.

9.9 Solid Edge Interface

Importing Solid Edge Files

Solid Edge uses Parasolid as its geometry engine. This interface extracts the existing Parasolid geometry from the Solid Edge file and loads it, just like loading a native Parasolid (X_T) file. This makes the conversion very robust.

To use the FEMAP Solid Edge interface, select the file that you want, and it will be loaded into FEMAP as Parasolid geometry.

If there is an existing solid, the interface will ask you if you want to update it. If you answer *Yes*, FEMAP will display the *Geometry Replacement Options* dialog box. For more information on this dialog box, see Section 9.2, "Parasolid Interfaces (*.X_T Format)"

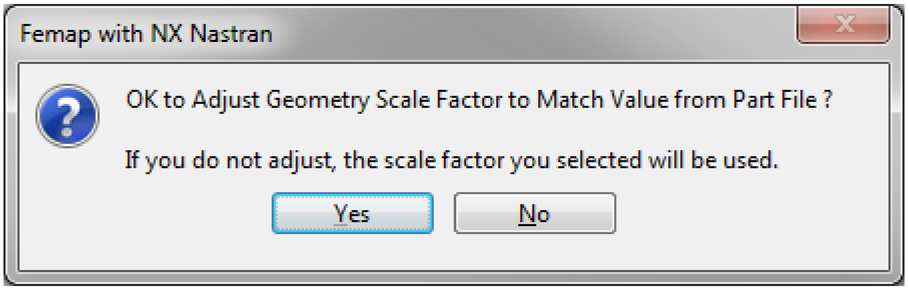
The Solid Edge interface can read part (.PAR), sheet metal (.PSM), legacy weldment (.PWD), and assembly (.ASM) files. Likewise, it only transfers the current geometry. Feature and other attribute information is not supported.

Note: As an alternative to directly reading the Solid Edge part file, you can use Solid Edge to export a Parasolid (X_T) file which FEMAP can read. In Solid Edge, you will typically use the *File, Save As* command, set the *File Type* to *Parasolid,* and write your part.

Using the FEMAP Add-in inside Solid Edge

When the Applications, Simulation, FEMAP command is used in Solid Edge V18 and above, the Solid Edge part, sheet metal part, legacy weldment, or assembly will be automatically sent to FEMAP. A new instance of FEMAP will be opened, if one is not currently open, and the part will be imported.

During import, FEMAP will first ask “OK to Adjust Geometry Scale Factor to Match Part? If you do not adjust the scale factor, geometry will import at the wrong size.” Clicking *Yes* will adjust the scale factor to the one which has been set in Solid Edge, while clicking *No* could result in your model being imported into FEMAP in the incorrect scale. Once the scale factor has been accepted or declined, the geometry will be read into FEMAP



If your part or assembly contains multiple part representations (i.e. a Design Model and a Simplified Model), FEMAP will ask which representation you would like to read into FEMAP. Depending on whether you are trying to read in a part or assembly may dictate which option you will want to use.

For a single part, choosing "Design Only" will read in the Design Model only, choosing "Simplified Only" will only read the simplified part if one exists, and choosing "All" will read in both the Design and Simplified Models. The "Simplified or Design" option will read in a Simplified Model if there is one for the part, but if not, will read in the Design Model.

For assemblies, these options work pretty much the same, except "Design Only" will only read in Design Models, so if a Simplified Model has been used in the assembly, it will not be read into FEMAP. The opposite is true if "Simplified Only" is selected, as only Simplified Models will be imported. "Simplified or Design" will scan the assembly and import a Simplified Model if one exists, but if not, the Design Model will be read in instead. "Simplified and Design" is the default, as it will read in the most simplified assembly available, while assuring that a representation will be read in for each part in the assembly.

9.10 NX Interface

NX is a high end integrated CAD/CAM software from Siemens Product Lifecycle Management Software, Inc., which also develops FEMAP. NX is based on Parasolid geometry kernel. As such, the FEMAP NX interface simply extracts the existing Parasolid geometry out of the file and loads it into FEMAP. No conversion is required because both systems are using the same geometry engine. The translator also supports files from Unigraphics versions 11 to 18.

To import a NX file, choose *File, Import, Geometry* and choose the appropriate filename. Since Pro/E also uses the .PRT extension, when you have the File Type set to *All Geometry*, FEMAP attempts to automatically distinguish and choose the right interface to read the file that you select. If it cannot determine the type automatically, it will ask you to specify the type. If you are having problems with this, open the *File Type* list and directly choose the *NX* interface.

9.11 SolidWorks Interface

SolidWorks uses Parasolid as its geometry engine. This interface extracts the existing Parasolid geometry from the SolidWorks file and loads it, just like loading a native Parasolid (X_T) file. This makes the conversion very robust.

To use the FEMAP SolidWorks interface, select the file that you want, and it will be loaded into FEMAP as Parasolid geometry.

If there is an existing solid, the interface will ask you if you want to update it. If you answer *Yes*, FEMAP will display the *Geometry Replacement Options* dialog box. For more information on this dialog box, see Section 9.2.1, "Reading Parasolid (X_T) Files".

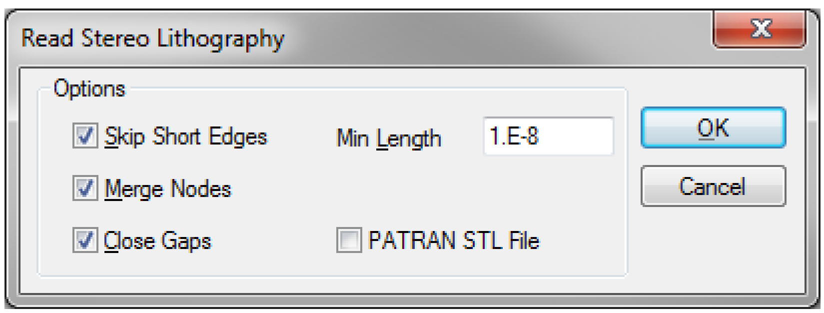
The SolidWorks interface can read part (.SLDPRT) and assembly (.SLDASM) files. Likewise, it only transfers the current geometry. Feature and other attribute information is not supported.

Note: As an alternative to directly reading the SolidWorks part file, you can use SolidWorks to export a Parasolid (X_T) file which FEMAP can read.

9.12 Stereolithography Interface

In addition to traditional CAD data, FEMAP can also read files that were generated for StereoLithography production. These files are usually text files, but can be binary as well. FEMAP can read all text versions, and binary files if they were generated on a PC. The facets in the StereoLithography file are converted to triangular surface elements during the translation.

When you select the *File, Import, Geometry* command and select a Stereolithography file, you will see the *Read Stereo Lithography* dialog box.



In addition to just translating the triangular facets, however, FEMAP can automatically merge all coincident points and split any facets that are necessary to eliminate free edges in the mesh. The options on this dialog box control merging, definition of short edges, as well as closing of gaps. This results in a valid finite element mesh, although typically with very bad aspect ratio elements. The facets can be remeshed with the *Mesh, Remesh* commands.

Use the *PATRAN STL File* option to read PATRAN STL files. This option will also let you read any STL file where the negative coordinate locations are written without spaces between the X,Y, and Z components.

Writing Stereolithography Files

FEMAP can also write Stereolithography files as well. FEMAP will ask you to select the elements to be translated. You will usually want to pick plane element types for this command, solids may be selected as well.. If you have quadrilateral plane elements, FEMAP will still write the triangular facets by first splitting these quads. This splitting procedure does not modify the elements themselves. FEMAP simply writes the triangular results to the STL file. If you choose solid elements, FEMAP will write each face as a triangular facet, splitting quadrilateral faces when required. If any plane elements are coincident with the face of solid elements, only one of these faces will be exported. Any line elements that are chosen in this procedure are simply ignored.

Customization 10

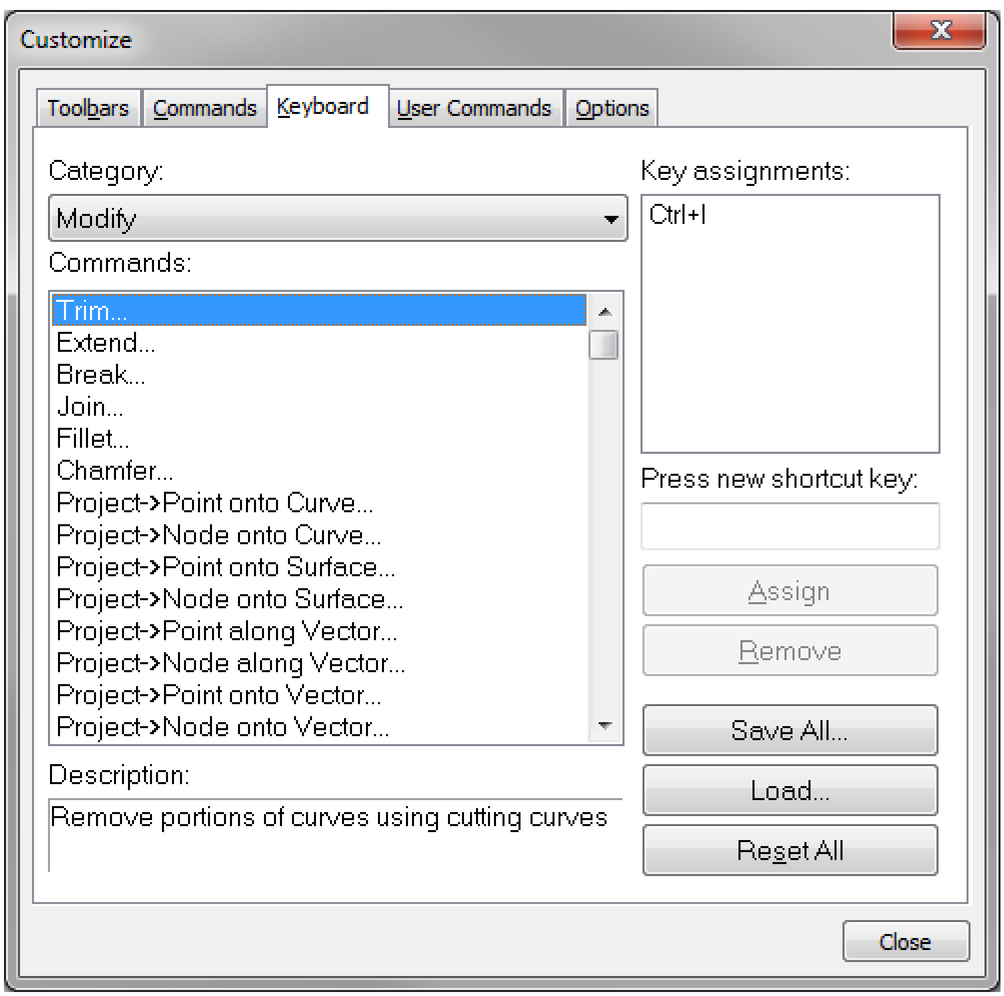
This topic contains information on customizing FEMAP. There are three basic sections:

- The first section describes the use of shortcut keys in FEMAP. You can define any letter, function key, or keyboard combination (i.e., CTRL, SHIFT, ALT + letter or function keys) to be a FEMAP command. This option enables you to quickly access your most commonly used commands from your keyboard.
- The second section describes customizing the Toolbars. You can create your own toolbars and add whatever existing or user commands you would like to a toolbar by dragging and dropping icons and menus onto a blank or existing toolbar. You can also to toggle certain icons on and off temporarily from toolbars as well.
- The third section briefly describes the FEMAP Application Programming Interface (API) and how it can be used to create user commands. There is much more information about the API in the *FEMAP API Reference* document included on the FEMAP CD.

A description of each of these customizable options is provided below.

10.1 FEMAP Shortcut Keys

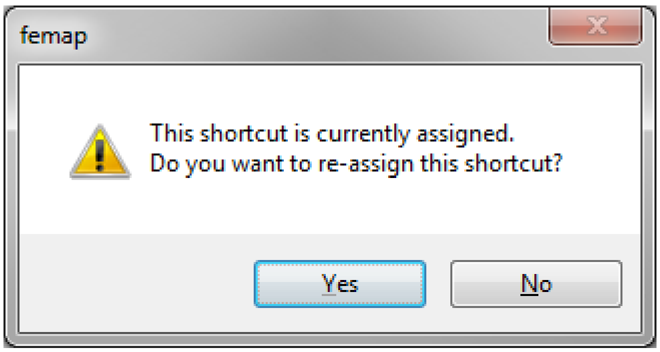
FEMAP has both certain keys defined as commands for quick implementation as well as providing you the capability to define your own shortcut keys. Commands which can be accessed through standard shortcut keys have the shortcut key listed next to their name. Some of the most commonly used shortcut keys include *F5* for *View Select*, *F6* for *View Options*, and *Ctrl+Q* for *View, Visibility*. These shortcut keys enable you to access these commands without going through the menu substructure.



In addition to the standard shortcut keys, FEMAP also allows you to define letter keys in FEMAP as FEMAP commands. You can also assign currently unused function keys and keyboard combinations (i.e., CTRL, SHIFT, ALT + letter or function keys) as FEMAP commands as well. You can therefore quickly customize FEMAP to use letter and function keystrokes, as well as keyboard combinations, to represent your most often used FEMAP commands

To set up your own shortcut keys, click the “customize” triangle on any toolbar and choose the *Customize...* command or use the *Tools, Toolbars, Customize...* menu. In both cases, the *Customize...* command is at the bottom of the menu. Once in *Customize* dialog box, choose the *Keyboard* tab.

To define a shortcut key, first choose the *Category* from the drop down list, then highlight the command from the *Commands* list. After the command is highlighted, click in the “Press new shortcut key:” field and press a key or keyboard combination. Once you have chosen the correct key or keyboard combination, click the *Assign* button.



If the key or keyboard combination has already been defined, FEMAP will let you know and bring up a dialog box stating “This shortcut is currently assigned. Do you want to re-assign this shortcut?” By clicking the *Yes* button, the key or keyboard combination will be added to the “Key assignments:” list and REMOVED from the command that was previously using that shortcut key or keyboard combination. Clicking the *No* button allows you to select an unused shortcut key or keyboard combination and leaves all other shortcut keys unchanged.

Shortcut keys can be saved by clicking the *Save All* button. FEMAP will prompt you to create a “Keyboard Shortcut File” (*.KEY file). This file will contain all of the keyboard shortcuts you have currently set in FEMAP. You can then click the *Load* button to load a *.KEY file and your shortcuts will be restored. For FEMAP versions 9.3 and above, you can load a *.KEY file from the previous version and quickly customize the new version.

Shortcut keys can be manually removed by highlighting a key or keyboard combination from the “Key Assignments:” list and then clicking the *Remove* button. The *Reset All* button will return all shortcut keys to their default commands.

Defining shortcut keys for your most used commands, you can save time moving through the FEMAP menu structure. Shortcut keys are only available from the FEMAP menu level. If you are already in another command or dialog box, pressing these keys will not have the desired effect. In most cases, it will simply result in typing the letter that you pressed

See Section 4.2.2.2, "Customizing toolbars" for some more information on creating shortcut keys.

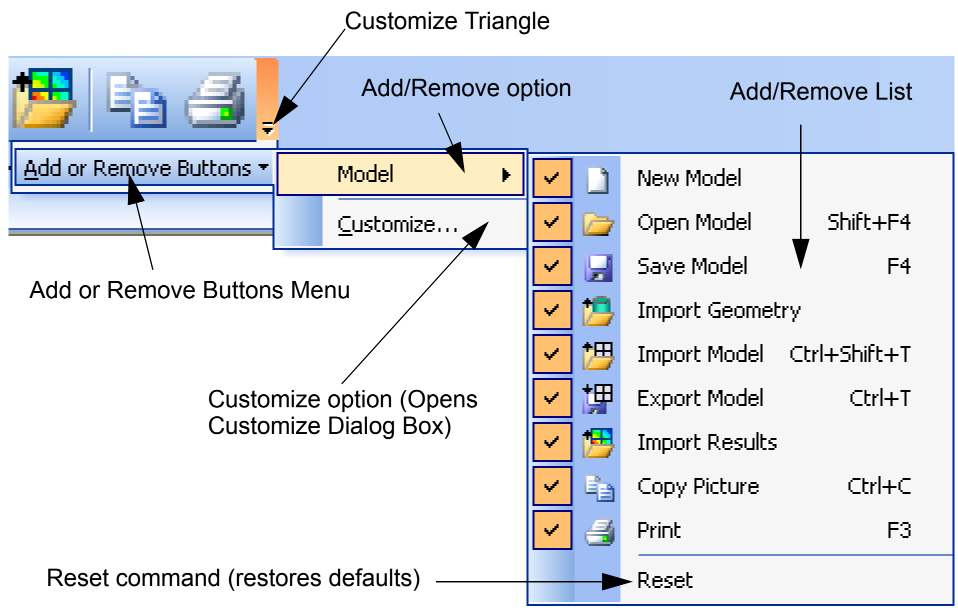
A few of the more useful but less obvious shortcut keys are listed below. These keys work within a text or drop down list box in a FEMAP dialog box or list boxes in FEMAP. They do not apply to other Windows applications except for those noted as Windows commands. For a complete list of shortcut keys, see Section A, "Using the Keyboard".

Key(s)	Function
Ctrl+A	Measure an angle.
Ctrl+C	Copy (Windows command)
Ctrl+D	Measure a distance.
Ctrl+E	Display FEMAP Equation Editor for interactive definition of variables and equations.
Ctrl+F	List functions.
Ctrl+G	Snap cursor selections to snap grid.
Ctrl+L	Display a list of the existing entities of the desired type.
Ctrl+N	Snap cursor selections to nearest node.
Ctrl+P	Snap cursor selections to nearest point.
Ctrl+S	Snap cursor selections to screen (snap off).

Ctrl+T	Redefine snap grid.
Ctrl+V	Paste (Windows command)
Ctrl+W	Redefine workplane.
Ctrl+X	Cut (Windows command)
Ctrl+Z	Use standard coordinate selection dialog box to define location.

10.2 Customizing Toolbars

FEMAP gives you the ability to customize the toolbars in several different ways. All customization begins by clicking on the small triangle (“Customize” triangle) that is on every visible toolbar (It appears in a different place depending on whether the toolbar is docked or floating). When the “Customize” triangle is clicked, a menu will drop down which says “Add or Remove Buttons”. When the “Add or Remove Buttons” menu is highlighted, it will bring up a second level menu with two options, Add/Remove from current toolbar or “Customize”.



Add/Remove option

The add/remove option will show as the current toolbar name, which when highlighted will bring up another menu level which allows you to individually turn existing icons on or off (You can turn multiple icons on or off while the menu is open and the toolbar will dynamically change). When the icon and command name have a check mark next to them, the icon is visible on the toolbar. To restore the default settings for a toolbar, choose Reset at the bottom of the menu.

Customize... option

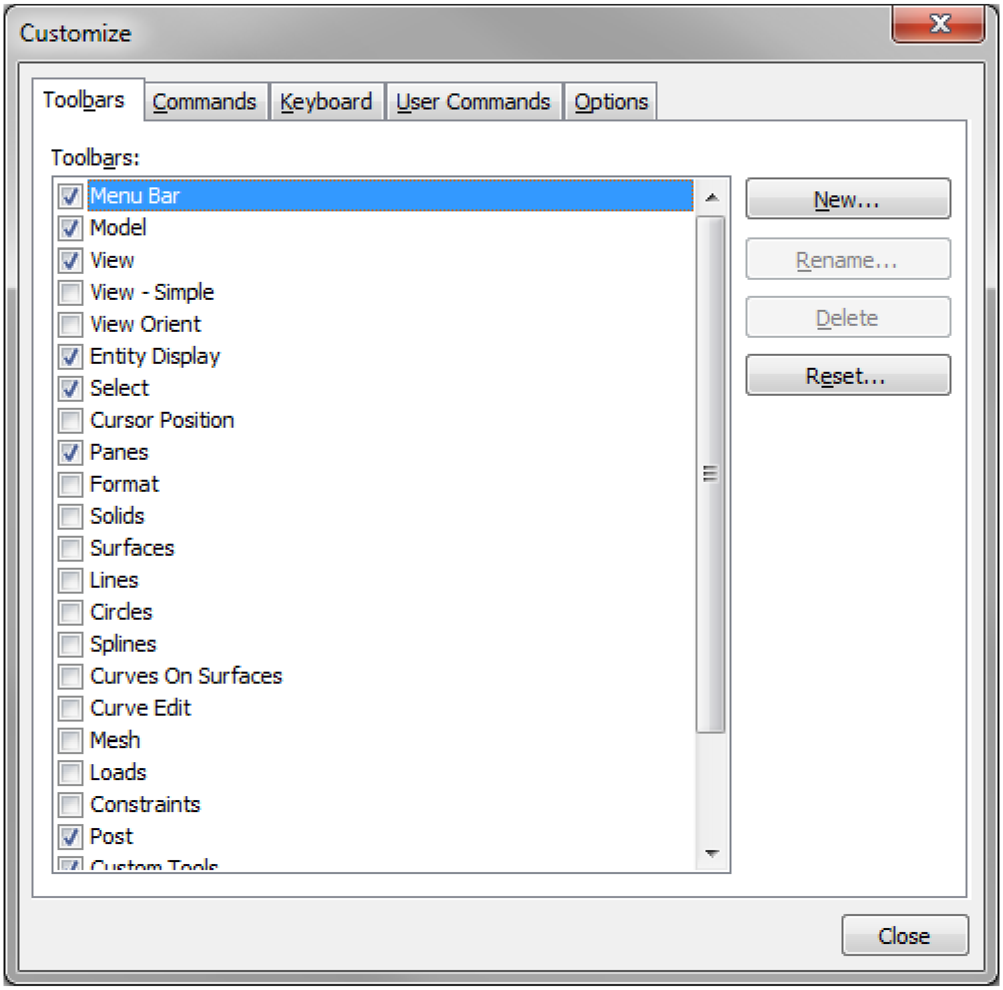
The Customize... option will bring up the Customize dialog box when clicked. Once open, this dialog box contains five different tabs which represent various methods to customize your toolbars. Also, while the Customize dialog box is open, you can right mouse click on any icon in any visible toolbar and a “Customize Icon” menu will appear. We will discuss the Customize dialog box and Customize Icon menu in greater detail below.

Customize Dialog Box

...The Customize Dialog box is broken into five different sections: Toolbars, Commands, Keyboard, User Commands, and Options. Each of these sections pertains to a specific area of toolbar customization. There is a tab for each heading that can be clicked to bring up the specific options for each section.

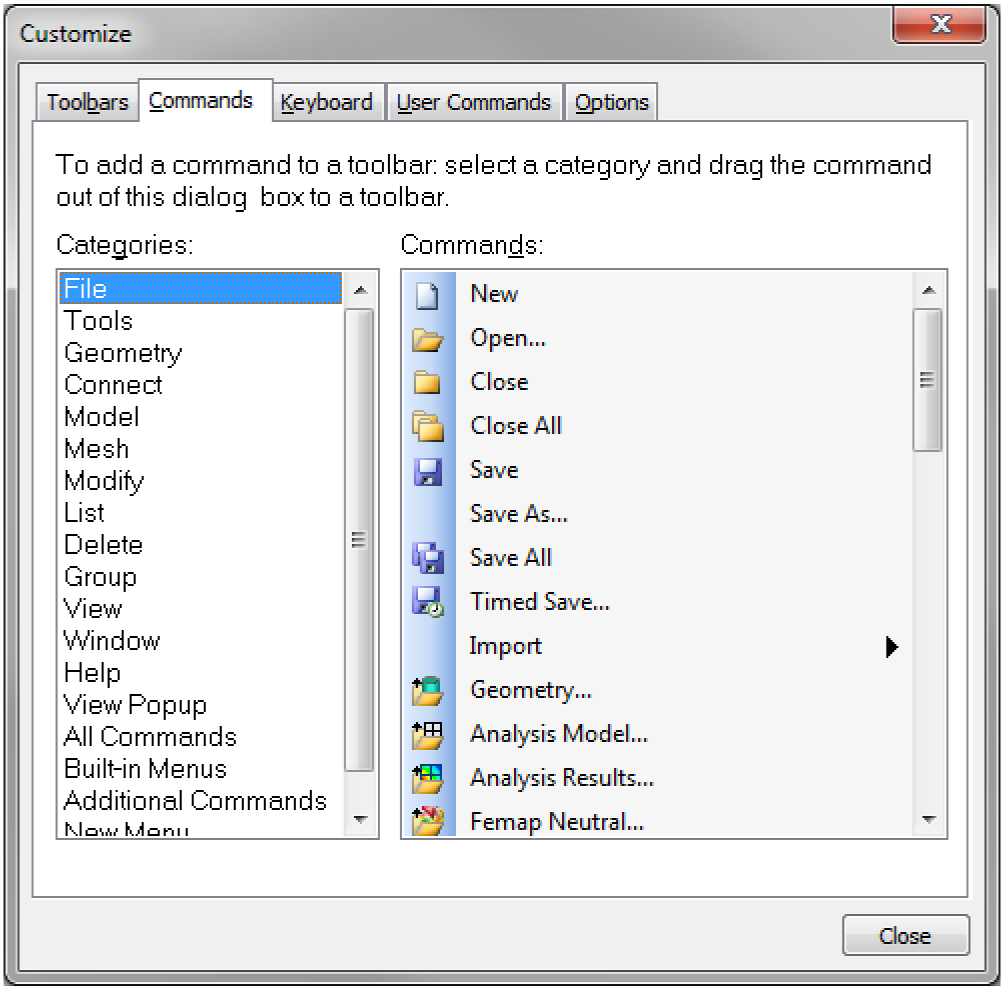
Toolbars

...Allows you to turn toolbars on and off by clicking the check box next to the toolbar name. This allows you to turn multiple toolbars on and off while in the same command. As each toolbar is checked or unchecked, it will appear or disappear in the FEMAP interface. This tab also allows you to create new, personalized toolbars by pressing the *New* button. FEMAP will prompt you to give the new toolbar a name and will bring up a “blank” toolbar in the FEMAP interface, which you can then add icons for exiting commands or user commands. “Personalized” toolbars can be renamed at any time using the *Rename* button or deleted using the *Delete* button. Using the *Reset* button will reset the toolbar highlighted in the list to the default configuration.

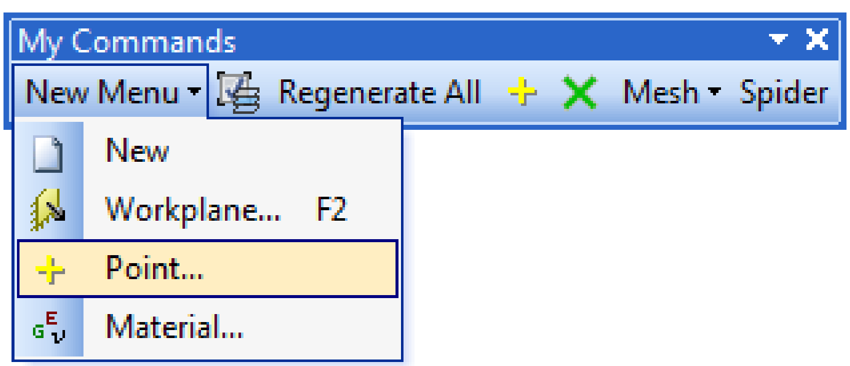


Commands

...The *Commands* tab contains all the commands available in FEMAP through the Main Menu structure. Choose the type of command you are looking for from the *Categories* list, then locate the specific command in the *Commands* list. Once the specific command is located, click and hold the left mouse button to “grab” the command. Now you can drag the “grabbed” command onto a visible toolbar and place it on that toolbar. Along with the commands available through the Main Menu structure, categories such as “Additional Commands” and “View Popup” allow access to specific view options and “right mouse menu” selections. You may also add an entire existing FEMAP menu to a toolbar using the “Built-in Menus” category or create a new menu of existing and user commands by dragging the New Menu command onto a toolbar and then filling the blank menu with commands. Any user commands will show up in the “User Commands” category. Any combination of icons and commands can be put together on a “personalized” toolbar.



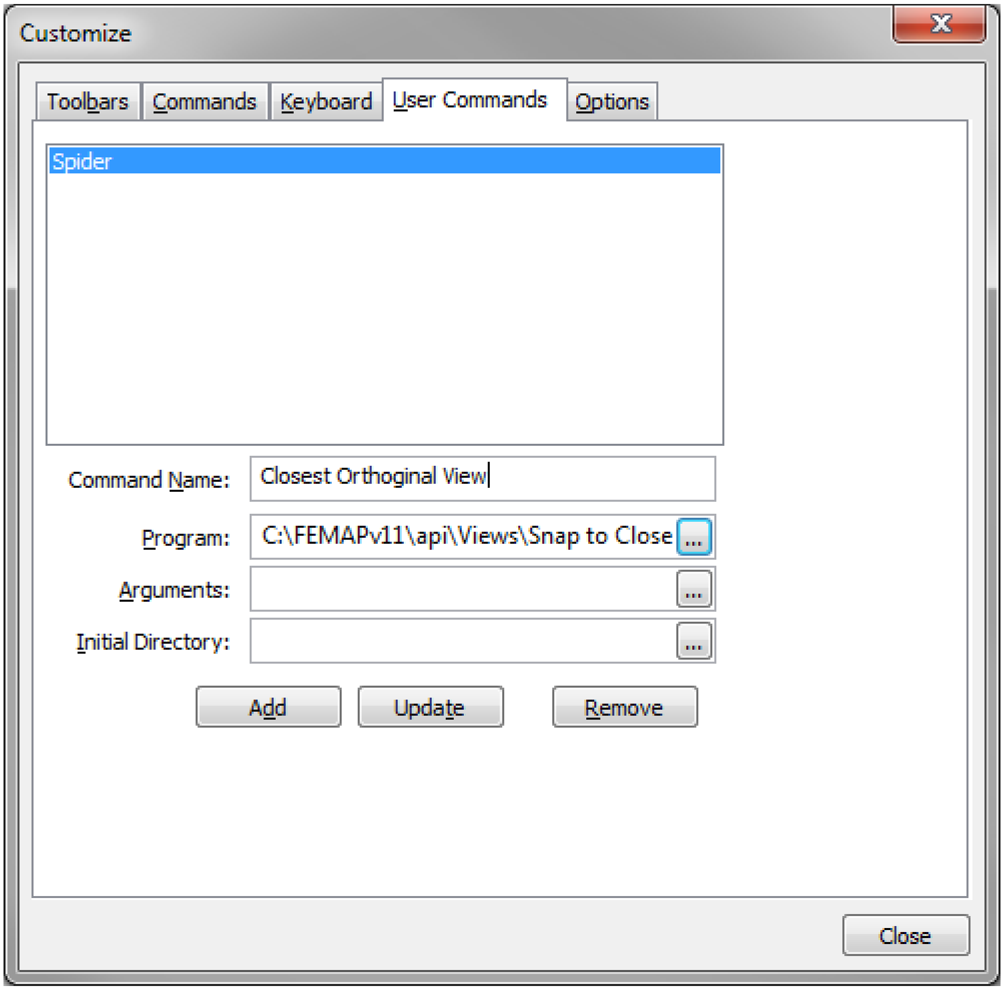
Many commands have icons which do not appear on any existing standard toolbar. These icons are in FEMAP specifically so you can add commands to existing toolbars and create your own "personalized" toolbars.



An example of a "personalized" toolbar can be seen to the left. Notice that there is a "New Menu" containing a few existing commands from different menus and toolbars that appear on a drop-down menu. Also included on this Custom toolbar are the *Visibility* icon from the "View" category, *View Regenerate All* command from the "Additional Commands" category, the *Snap to Point* and *Snap to Node* icons from the "View Popup" category, the entire *Mesh* menu from the "Built-in Menus" category, and *Spider* (a user command) from the "User Commands" category.

User Commands

...The *User Commands* tab allows you to create command names for user commands created using the FEMAP Applications Programming Interface (API).



In order to locate a file to be used as a program, you can browse through windows directories using the "..." browse button next to the *Program* field. Choose the file to be used as the "program" file, click OK, and then the entire directory path will be shown in the *Program* field. There are several different files which can be used as a "Program" files including Executable (*.exe), Command (*.com), Information (*.pif), and Batch (*.bat, *.cmd) files

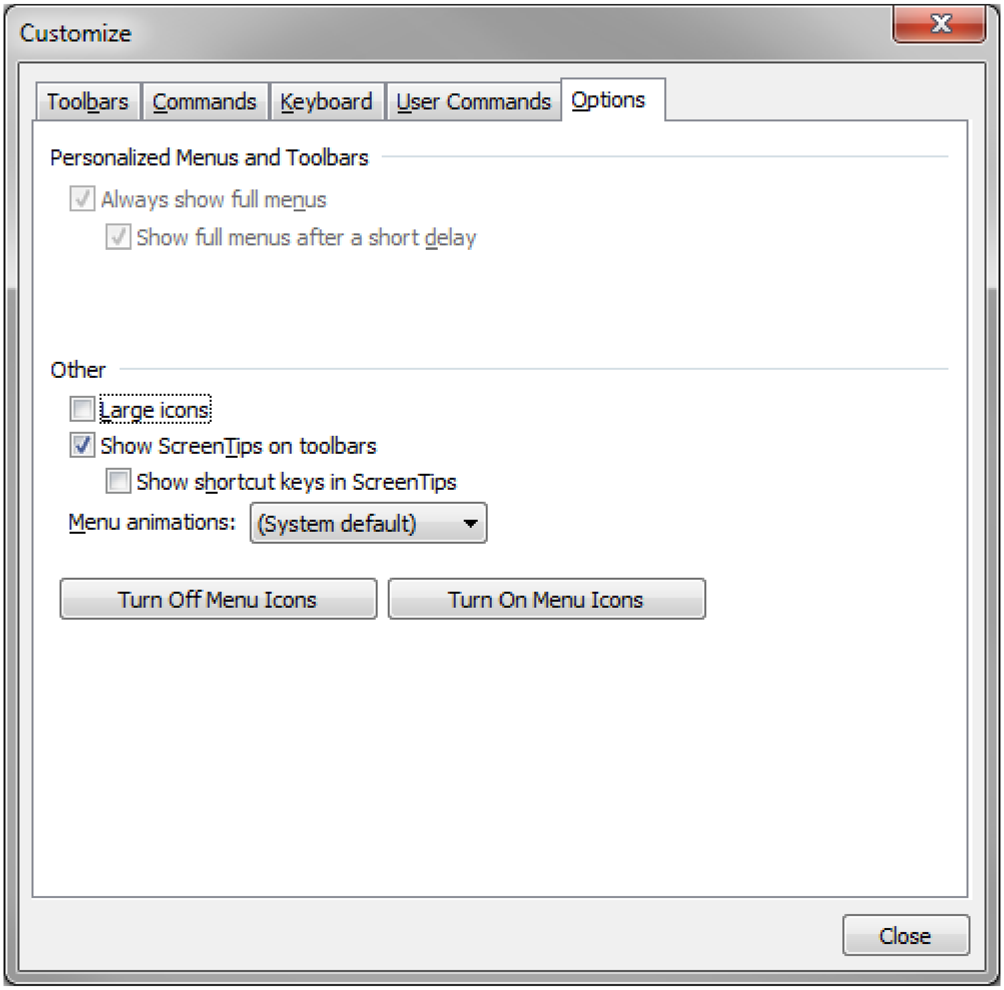
Once the file for the actual command has been located, the command must be given a unique Command Name. After the command has been given a name, click the *Add* button to place it into the list of User Commands. If you would like to change the name or directory path of a User Command, highlight it in the list, make any modifications, then click the *Update* button to confirm the change. To remove a User Command from the list, highlight it, then click the *Remove* button.

Along with the "Program" file itself, you may optionally enter other necessary files and command line entries into the *Arguments* field. In addition, if any program file needs to use an external directory, the path to that directory can be entered into the *Initial Directory* field.

Once the commands and are added to the User commands list, they will appear in the "User Commands" category in both the *Commands* and *Keyboard* sections of the Customize dialog box. User commands can now be added to existing toolbars or "Personalized" toolbars using the methods described in the *Toolbars* and *Commands* sections.

Options

....Allows you to select options to make the toolbars more useful. At the current time, the “Personalized Menus and Toolbars” options in the Options tab have no effect on any existing or custom FEMAP menus or toolbars. These options will be available in future versions.

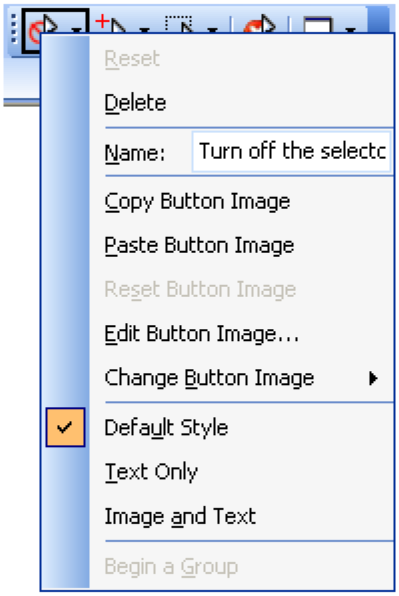


To make the icons on all the toolbars larger, select the “Large icons” option.

By default, the “Show ScreenTips on toolbars” option is on, you can uncheck the box to turn the ScreenTips off. If you would like the ScreenTips to also show all associated shortcut keys, use the “Show shortcut keys in Screen-Tips” option.

You can select the style of how the menus drop-down by selecting a style from the drop-down “Menu animations” list. The options are (System default), Unfold, Slide, Fade, or None for a particular style or choose Random, for a different “drop down” style each time.

Customize Icon menu



...The Customize Icon Menu is available only when the *Customize* dialog box is open. In order to use the commands on the Customize Icon Menu, right mouse click any icon on any visible toolbar. Only the icon that you selected will be altered by the commands on the Customize Icon Menu. This menu contains commands used to delete icons from a toolbar, reset the default icon, and change the name of an icon. It also allows you to copy, paste, reset, edit, or change the button image of an icon. Along with these functions, icon style can be selected, and icons can be separated into "groups" on toolbars using partitions.

A brief description of the commands on the Customize Icon Menu:

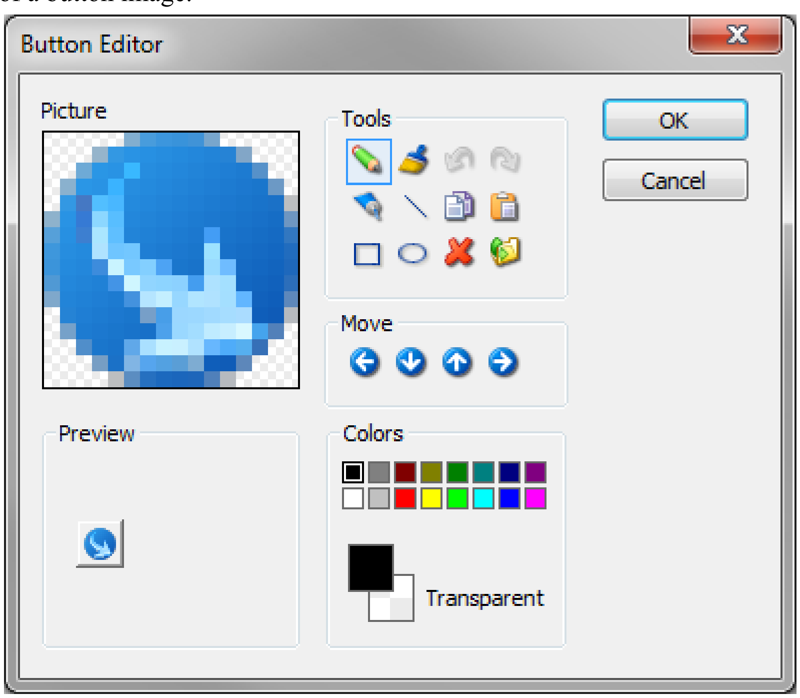
•**Reset:** Resets all icon options (name, button image, style, group) to default values.

•**Delete:** Removes icon from the toolbar it is currently on. If the icon appears on multiple toolbars, it will only be deleted from the toolbar that you initially right mouse clicked to open the Customize Icon Menu.

•**Name:** Allows you to change the name of an icon. This name will appear on the toolbar when the Icon style is set to Text Only or Image and Text

•**Copy Button Image:** Copies the button image to the clipboard.

- **Paste Button Image:** Once an icon image is on the clipboard, it can be pasted onto to another icon to replace that icon's current image.
- **Reset Button Image:** Resets the button image to the default button image.
- **Edit Button Image:** Brings up the *Button Editor* dialog box. In this dialog box, there are many tools to alter the appearance of a button image.



Button Image size is limited to a 16 X 16 square "picture". The existing picture can be modified by changing the colors or moving the image, a new picture can be drawn, a copied button image can be pasted in, or a picture from a file can be imported. Any combination of these methods can be used to create custom icons. There is a preview window that dynamically changes as you modify the icon and Undo and Redo tools to help modification. Once the image is finished, it can be copied to the clipboard as well.

Note: Any imported image will be reduced to a 16 X 16 pixel resolution image, so be sure to inspect all imported images to make sure they still resemble the image after the resolution reduction.

- **Change Button Image:** Allows you to choose a button image from a set of images provided by FEMAP.
- **Default Style:** Resets the icon style to the default setting. (Usually Button Image only)
- **Text Only:** Shows Icon Name only (no Button Image)
- **Image and Text:** Shows both the Button Image and the Icon Name together. (View Orient toolbar default)
- **Begin a Group:** When checked, creates toolbar partition line to the left (horizontal toolbars) or above (vertical toolbars) the icon being customized.

10.3 Introduction to the FEMAP API

FEMAP provides a robust set of finite element modeling and post-processing functionality. At times, however, you may need a specific capability that is not included in the standard product. The FEMAP Application Programming Interface (API) lets you customize FEMAP to meet your specific needs.

The FEMAP API is an OLE/COM-based programming interface to FEMAP. It contains hundreds of functions that can be called from *API Programming* Dockable Pane in FEMAP or other applications such as Visual Basic, VBA (Excel, Word, Access...), C, or C++. The *FEMAP API Reference* documents the objects and functions that are available in the FEMAP API.

For More information on the *API Programming* Pane, see Section 7.2.10, "Tools, Programming, API Programming" in the *FEMAP Commands Manual*

To use the FEMAP API successfully, you need to understand:

- Objects, Methods, and Properties (See Objects, Methods and Properties)
- Data Types (See Data Types)
- Memory Allocation (See Memory Allocation)
- Global Constants (See Global Constants)

For examples of how to use the FEMAP API, see the *FEMAP API Reference Manual*

Using the Keyboard

A

Command Keys

These keys provide shortcut methods for accessing the FEMAP commands. In addition to the keys described here, menu commands can also be activated by using the *Alt* key in combination with the underlined letter in the command (or by selecting them with the mouse). Many of these shortcuts are shown on the right side of the menus.

Key(s)	Command
Ctrl+A	*View, Autoscale, Visible*
Ctrl+Shift+A	*View, Autoscale, Regenerate All*
Ctrl+B	*Mesh, Between*
Ctrl+C	*File, Picture, Copy*
Ctrl+Shift+C	*File, Message, Copy*
Ctrl+D	*Window, Redraw*
Ctrl+E	*Model, Element*
Ctrl+F	*Modify, Fillet*
Ctrl+G	*Window, Regenerate*
Ctrl+H	*Help*
Ctrl+I	*Modify, Trim*
Ctrl+J	*Modify, Join*
Ctrl+K	*Modify, Break*
Ctrl+L	*Tools, Variable*
Ctrl+M	*View, Magnify*
Ctrl+N	*Model, Node*
Ctrl+O	*View, Options*
Ctrl+P	*View, Pan*
Ctrl+Shift+P	*File, Preferences*
Ctrl+Q	*View, Visibility*
Ctrl+R	*View, Rotate*
Ctrl+S	*View, Select*
Ctrl+T	*File, Export, Analysis Model*
Ctrl+Shift+T	*File, Import, Analysis Model*
Ctrl+U	Docks/undocks *Messages* window
Ctrl+Shift+U	Open/close Graphics Window
Ctrl+Y	Last menu command
Ctrl+W	*Tools, Workplane*
Ctrl+Z	*Tools, Undo*
Ctrl+Shift+Z	*Tools, Redo*
Ctrl+Ins	*File, Picture, Copy*
Ctrl+Alt+Ins	*File, Message, Copy*
Alt+Backspace	*Tools, Undo*
Alt+Shift+Backspace	*Tools, Redo*

Function Keys

These keys provide additional shortcut methods for accessing the FEMAP commands. In addition to the keys described here, menu commands can also be activated by using the *Alt* key in combination with the underlined letter in the command (or by selecting them with the mouse), or through various *Ctrl*-key combinations.

Key(s)	Command
F1	*Help*
F2	*Workplane*
Ctrl+F2	*Model, Load, Create/Manage Set*
Shift+F2	*Model, Constraint, Create/Manage Set*
Alt+F2	*Group, Create/Manage*
F3	*Print*
Ctrl+F3	*Save, Picture*
Shift+F3	*Page Setup*
Alt+F3	*Replay, Picture*
F4	*File, Save*
Ctrl+F4	*Window, Close*
Shift+F4	*File, Open*
Alt+F4	*File, Exit*
F5	*View, Select*
Ctrl+F5	*View, Visibility*
Shift+F5	*View, Post Data*
Alt+F5	*View, XY Data*
F6	*View, Options*
Shift+F6	*View, Visibility*
F7	*View, Zoom*
Ctrl+F7	*View, Magnify*
Shift+F7	*View, Autoscale, All*
Alt+F7	*View, All Views*
F8	*View, Rotate, Model*
Ctrl+F8	*View, Align by, Along Vector*
Shift+F8	*View, Center*
Alt+F8	*View, Pan*
F9	*Model, Line Coordinates*
Ctrl+F9	*Model, Arc, Points*
Shift+F9	*Model, Surface, Edge Curves*
Alt+F9	*Model, Volume, Surfaces*
F10	View menu bar
Shift+F10	*Mesh, Size Along Curve*
Alt+F10	Last menu command
F11	*Mesh, Between*
Ctrl+F11	*Mesh, Transition*
Shift+F11	*Mesh, Geometry, Surface*
Alt+F11	*Geometry, Boundary Surface, From Curves*
F12	*Window, Redraw*
Shift+F12	*Window, Show Entities*
Alt+F12	*Mesh, Geometry, Solid*

Dialog Box Keys

The following keys are implemented by Windows and are typical for most “well-behaved” Windows applications. They allow you to move around the dialog box, select options, enter and modify data, and execute additional commands.

Movement	
Key(s)	**Function**
Tab	Moves to next list box, text box, check box, command button, or group of option buttons. Moves from field to field (left to right and top to bottom).
Shift+Tab	Moves from field to field in reverse order.
Alt+letter	Moves to the option or group whose underlined letter matches the one you type.
Letter key	Moves to next item beginning with that letter in an active list or drop-down list box.
Direction key (Up, Down, Right, Left)	Moves from option to option within a group of check boxes, command buttons, or option buttons. Also moves selection in a normal or drop-down list box.
Selection or Operation	
Key(s)	**Function**
Enter	Executes the currently active command button. This is typically the *OK* button when the dialog box is first displayed. The currently active command button has a wider, darker border than the other buttons.
Esc	Closes a dialog box without completing the command. (Same as clicking the *Cancel* button.)
Alt+Down	Opens a drop-down list box.
Space (SpaceBar)	Turns on or off active check box or option button, or chooses the active command button.
Editing	
Key(s)	**Function**
Left or Right	Moves one character.
Home	Moves to beginning of line.
End	Moves to end of line.
Ins	Turns on/off overtype.
Shift+Direction key	Extends selection in a text box.
Shift+Home	Extends selection to first character in a text box.
Shift+End	Extends selection to last character in a text box.
Alt+Backspace	Single level undo/redo of the changes made to the active text box or drop-down list box.
Ctrl+Enter	Add a new line in a multi-line text entry control.
Editing with no characters selected	
Key(s)	**Function**
Del	Deletes character to right of insertion point.
Ctrl+Del	Deletes to end of line.
Ctrl+Shift+Del	Cuts to end of line.
Shift+Del	Cuts character to right.
Backspace	Deletes character to left of insertion point.
Shift+Ins	Pastes Clipboard data.
Editing with characters selected	
Key(s)	**Function**
Del	Deletes selection.
Ctrl+Del	Deletes from beginning of selection to end of line.

Ctrl+Shift+Del	Cuts from beginning of selection to end of line
Shift+Del	Cuts selection to Clipboard.
Backspace	Deletes selection.
Ctrl+Ins	Copies selection to Clipboard.
Shift+Ins	Replaces selection with Clipboard data.

Special Dialog and Toolbox Field Keys

The following keys work within text or drop-down list boxes in FEMAP. A * at the end of the description in the *Function* column below indicates the keys do NOT work in any fields found in the various Toolbox-style Dockable Panes. They do not apply to other Windows applications except for *Ctrl+X*, *Ctrl+C*, and *Ctrl+V*..

Key(s)	Function
Ctrl+A	Measure an angle
Ctrl+C	Copy (Windows command).
Ctrl+D	Measure a distance
Ctrl+E	Display FEMAP Equation Editor for interactive definition of variables and equations.*
Ctrl+F	*List, Functions**
Ctrl+G	Snap cursor selections to snap grid.
Ctrl+I	Measure the radius of a circular arc
Ctrl+L	Display a list of the existing entities of the desired type.*
Ctrl+M	Measure the length of a selected curve
Ctrl+N	Snap cursor selections to nearest node
Ctrl+P	Snap cursor selections to nearest point
Ctrl+R	Enable Smart Snap, which snaps cursor selections to the nearest point, node, midpoint of a curve, or center point of a circular arc
Ctrl+S	Snap cursor selections to screen (snap off)
Ctrl+T	Redefine snap grid.
Ctrl+V	Paste (Windows command).
Ctrl+W	Redefine workplane.
Ctrl+X	Cut (Windows command).
Ctrl+Z	Use standard coordinate selection dialog box to define location.*

Dialog Function Keys

These keys provide additional shortcut methods for modifying the orientation and position of the model in the graphics window when a FEMAP dialog box is open. F1 brings up “context sensitive” help for the command that currently has the dialog box open.

Key(s)	Command
F1	*Context Sensitive Help*
F2	*Magnify Up 110%*
Shift+F2	*Magnify Up 150%*
Ctrl+F2	*Magnify Up 200%*
Alt+F2	*View, Autoscale, Visible*
F3	*Magnify Down 110%*
Shift+F3	*Magnify Down 150%*
Ctrl+F3	*Magnify Down 200%*
Alt+F3	*View, Autoscale, Visible*
F4	*Box Zoom*
F5	*Pan Left 10%*
Shift+F5	*Pan Left 25%*
Ctrl+F5	*Pan Left 50%*

Key(s)	Command
Alt+F5	*Pan Left 100%*
F6	*Pan Right 10%*
Shift+F6	*Pan Right 25%*
Ctrl+F6	*Pan Right 50%*
Alt+F6	*Pan Right 100%*
F7	*Pan Up 10%*
Shift+F7	*Pan Up 25%*
Ctrl+F7	*Pan Up 50%*
Alt+F7	*Pan Up 100%*
F8	*Pan Down 10%*
Shift+F8	*Pan Down 25%*
Ctrl+F8	*Pan Down 50%*
Alt+F8	*Pan Down 100%*
F9	*Rotation Direction Toggle (positive/negative)*
F10	*Rotate about X by 10 degrees (model axis)*
Ctrl+F10	*Rotate about X by 90 degrees (model axis)*
Shift+F10	*Rotate about X by 10 degrees (screen axis)*
Alt+F10	*Rotate about X by 90 degrees (screen axis)*
F11	*Rotate about Y by 10 degrees (model axis)*
Ctrl+F11	*Rotate about Y by 90 degrees (model axis)*
Shift+F11	*Rotate about Y by 10 degrees (screen axis)*
Alt+F11	*Rotate about Y by 90 degrees (screen axis)*
F12	*Rotate about Z by 10 degrees (model axis)*
Ctrl+F12	*Rotate about Z by 90 degrees (model axis)*
Shift+F12	*Rotate about Z by 10 degrees (screen axis)*
Alt+F12	*Rotate about Z by 90 degrees (screen axis)*

Menu Keys

The following keys allow you to access and move within the FEMAP menus. You can also access the menus using the mouse.

Key(s)	Function
Alt or F10	Activates menu bar.
Esc	Cancels menu.
Underlined letter	Displays menu.
Left or Right	Highlights the menu to the left or right.
With menu displayed	
Key(s)	**Function**
Underlined letter	Chooses command or sub-menu.
Enter	Chooses highlighted command.
Esc	Cancels menu.
Up	Highlights previous command.
Down	Highlights next command.
Left or Right	Displays the menu to the left or right.

Selection Keys

The following keys are used in FEMAP to assist in selection of multiple entities. They are used in combination with pressing the left mouse button and dragging the mouse.

Key(s)	Function
Ctrl	Select all entities within a circular area.
Shift	Select all entities within a square/rectangular area.
Ctrl+Shift	Select all entities within a polygon area.

Windows Keys

The following keys allow you to switch between applications, or between windows within a single application, and to scroll data within a window with scroll bars, such as the FEMAP *Messages* window.

Key(s)	Function
Alt+Esc	Next application
Alt+Shift+Esc	Previous application
Alt+Tab	Next windowed application
Alt+Shift+Tab	Previous windowed application
Ctrl+Esc	Display the Windows task list
In FEMAP Messages Window	
Key(s)	**Function**
Direction keys	Scroll in appropriate direction.
PgUp and *PgDn*	Scroll window up/down one screen.
Ctrl+PgUp	Places cursor in current, visible top line of *Messages* window
Ctrl+PgDn	Places cursor in current, visible bottom line of *Messages* window
Home	Beginning of line.
End	End of line.
Ctrl+Home	Scroll to first line of the *Messages* window
Ctrl+End	Scroll to last line of the *Messages* window

B Using the Mouse and Touch

At the FEMAP Menu Level - Mouse

While FEMAP is waiting for you to choose a command, you can perform certain “FEMAP-specific” functions with the mouse. These are in addition to the normal Windows functions that you can perform by clicking and dragging the mouse in the window borders and menus. All references to left and right are for a standard “right-handed” mouse configuration.

Action	Result
General	
Click button in window	Activate that window. If you have multiple graphics windows on the screen the one you click on will become active. All view changes will be made to that window.
Graphics Window	
Click, hold and drag left mouse button in Graphics window.	*Dynamic Rotate, Pan* and *Zoom*. Works identical to the *Dynamic Rotate, Pan, and Zoom* View Toolbar command, except you do not have to enter a command to do it.
Double click left mouse button in Graphics window.	Redraws that window.
Click, hold and drag middle mouse button or mouse wheel in Graphics window.	*Dynamic Rotate, Pan* and *Zoom*. Works identical to the *Dynamic Rotate, Pan, and Zoom* View Toolbar command, except you do not have to enter a command to do it.
Spin Mouse Wheel in Graphics Window	Zooms in and out. If the mouse wheel is spun with *Ctrl* key down, model will rotate around the screen X-axis; with *Shift* key down it will rotate around the screen Y-axis; with both *Ctrl* and *Shift* keys down at the same time, it will rotate around the screen Z-axis.
Double click middle mouse button or mouse wheel in Graphics window.	Autoscales to show all currently visible entities. If holding *Shift* key down, Autoscales to show all extents of all entities in the model, visible or not.
Click right mouse button in Graphics window (*Select Toolbar* not active)	Display quick access menu.
Click right mouse button in Graphics window (*Select Toolbar* active)	Displays Context Sensitive menu for the entity type which is currently active in the *Select Toolbar*.
Press *Shift* key, then press and drag left mouse button in a graphics window (*Select Toolbar* active).	Select all entities inside the square (rectangle) that is formed as you drag the mouse. This only works in the standard entity selection dialog box.
Press *Ctrl* key, then press and drag left mouse button in a graphics window (*Select Toolbar* active).	Select all entities inside the circle that is formed as you drag the mouse. This only works in the standard entity selection dialog box.
Press *Ctrl* + *Shift* key, then press left mouse button one time in a graphics window (*Select Toolbar* active).	Begins selection of all entities inside a polygon of any number of sides that is formed as you click the left mouse button. as many times as needed. Double-click of click *Done* in Polygon Picking dialog box to finish selection. This only works in the standard entity selection dialog box.
When model in animating, click left mouse button in the graphics window	Temporarily pause the animation, begins again when finger is removed from screen
Messages **window**	
Click left mouse button in *Messages* window.	Select the line that you were pointing at for transfer via *File, Messages, Copy* or *File, Messages, Save*. Also, press Shift + the left mouse button to select a range of lines.

Action	Result
Press and drag left mouse button in Messages window.	Select all lines between where you press and release the mouse button.
Click right mouse button in *Messages* window.	Brings up a Context Sensitive menu for the *Messages* window.
Double click left mouse button on *Messages* window Title Bar. (Docked or Floating)	Toggles the *Messages* window between Docked and Floating.
Double click left mouse button on *Messages* window Title Bar. (Docked and Tabbed, during "fly-out")	While the *Messages* window is "flown-out", will change the *Messages* window from Docked and Tabbed (Retracted) to simply Docked.
Entity Editor, Meshing Toolbox, and PostProcessing Toolbox	
Click left mouse button in any field or +/- toggle *Entity Editor*, *Meshing Toolbox*, or *PostProcessing Toolbox*.	Makes the field active for editing (except Read-Only fields). Clicking on the "Down arrow button" will bring up a drop-down menu and clicking on the "Action button" will bring up a dialog box. Also can be used to expand and collapse different sections by pressing the "+/-" toggle.
Click left mouse button on title bar of an individual in *Meshing Toolbox* or *PostProcessing Toolbox*.	Makes the selected tool the "active" tool.
Click left mouse on title bar of an individual in *Meshing Toolbox* or *PostProcessing Toolbox*.	Makes the selected tool the "active" tool.
Press and drag left mouse button in any field *Entity Editor*, *Meshing Toolbox*, or *PostProcessing Toolbox*.	Selects all characters in the field between where you press and release the mouse button.
Double click left mouse button on window Title Bar. (Docked or Floating)	Toggles the *Dockable Pane* between Docked and Floating.
Double click left mouse button on window Title Bar. (Docked and Tabbed, during "fly-out")	While the *Dockable Pane* window is "flown-out", will change the *Dockable Pane* window from Docked and Tabbed (Retracted) to simply Docked.
Data Table and Connection Editor)	
Click left mouse button on a row.	Selects and highlights the row the cursor is currently over. (Press Ctrl + the left mouse button to select multiple rows one at a time or Shift + the left mouse button to select a range of rows.)
Click left mouse button on a column heading.	Toggles the Sort Method from lowest to highest (numerical and alphabetical) and vice versa.
Click left mouse button on a column heading and drag the heading	Allows you to move the column headings around and place them in the order you wish. Also, dragging a column header off the row of column headers until an "X" appears over the column heading will remove the column from the *Dockable Pane*.
Click right mouse button on a row.	Brings up a Context Sensitive menu allowing you to perform show, filter, or delete the selected rows.
Click right mouse button on a column heading.	Brings up a Context Sensitive menu with sorting, alignment, and other options.
Double click left mouse button on window Title Bar. (Docked or Floating)	Toggles the *Dockable Pane* between Docked and Floating.
Double click left mouse button on window Title Bar. (Docked and Tabbed, during "fly-out")	While the *Dockable Pane* window is "flown-out", will change the *Dockable Pane* window from Docked and Tabbed (Retracted) to simply Docked
Model Info tree	

Action	Result
Click left mouse button in *Model Info* tree on an entity.	Selects and highlights the entity in the tree the cursor is currently over. Also can be used to expand and collapse the different branches of the “tree”, by pressing the “+/-” toggle and toggle visibility check boxes on/off.
Click right mouse button in *Model Info* tree on an entity.	Displays Context Sensitive menu for the entity which is currently highlighted in the *Model Info* tree.
Click right mouse button in *Model Info* tree below the “Selection List”.	Brings up a Context Sensitive menu with *New*, *Open*, and *Exit* options.
Double click left mouse button on *Model Info* window Title Bar. (Docked or Floating)	Toggles the *Model Info* tree between Docked and Floating.
Double click left mouse button on *Model Info* window Title Bar. (Docked and Tabbed, during “fly-out”)	While the *Model Info* tree window is “flown-out”, will change the *Model Info* tree from Docked and Tabbed (Retracted) to simply Docked
Status Bar	
Click left mouse button over an entity name in the “Tray” portion of the *Status Bar*.	Brings up a Context Sensitive Menu depending on the entity type the cursor is currently positioned over.
Click right mouse button anywhere on the *Status Bar*.	Brings up a Context Sensitive Menu which allows you to turn different entity types on and off in the “Tray”.

During a Command

When a command dialog box is displayed, the mouse can perform the following functions. These are also in addition to the normal Windows functions which allow you to move between dialog controls, and manage your windows.

Action	Result
Click right mouse button in graphics window.	Display quick access menu.
Click left mouse button in graphics window.	Graphically select the closest entity or location. This only works when you would normally be typing in a text, list or combo box which references a selectable entity.
Click, hold and drag middle mouse button or mouse wheel in Graphics window.	*Dynamic Rotate, Pan* and *Zoom*. Works identical to the *Dynamic Rotate, Pan, and Zoom* View Toolbar command, except you do not have to leave the open command to do it.
Spin Mouse Wheel in Graphics Window	Zooms in and out. If the mouse wheel is spun with *Ctrl* key down, model will rotate around the screen X-axis; with *Shift* key down it will rotate around the screen Y-axis; with both *Ctrl* and *Shift* keys down at the same time, it will rotate around the screen Z-axis.
Click middle mouse button or mouse wheel in graphics window.	When *Middle Button Click for OK* option is enabled on the *User Interface* tab of *File, Preferences*, presses OK on any open dialog box.
Press *Shift* key, then press and drag left mouse button in a graphics window.	Select all entities inside the square (rectangle) that is formed as you drag the mouse. This only works in the standard entity selection dialog box.
Press *Ctrl* key, then press and drag left mouse button in a graphics window.	Select all entities inside the circle that is formed as you drag the mouse. This only works in the standard entity selection dialog box.
Press *Ctrl* + *Shift* key, then press left mouse button one time in a graphics window.	Begins selection of all entities inside a polygon of any number of sides that is formed as you click the left mouse button. as many times as needed. Double-click of click *Done* in Polygon Picking dialog box to finish selection. This only works in the standard entity selection dialog box.

At the FEMAP Menu Level - Touch

While FEMAP is waiting for you to choose a command, you can perform certain “FEMAP-specific” functions by using touch gestures. These are in addition to the normal Windows functions that you can perform with touch gestures.

Action	Result
General	
Touch in window	Activate that window. If you have multiple graphics windows on the screen the one you touch will become active. All view changes will be made to that window.
Graphics Window	
Press, hold, and drag with one finger in the Graphics Window	Dynamically rotates the model
Press, hold, and drag with two or more fingers together in the Graphics Window	Dynamically pans (translates) the model up, down, left, or right with minimal rotation of the model.
Press with two fingers apart from one another and move fingers towards each other (works with more than two fingers, but easiest with only two)	Zooms out.
Press with two fingers together and move fingers away each other (works with more than two fingers, but easiest with only two)	Zooms in
Press with two fingers and move one finger “around” the other finger (fingers apart from one another is easiest)	Rotates around the “Z-Axis of the screen” using the location of the stationary finger.
When model in animating, press and hold any number of fingers in the graphics window	Temporarily pause the animation, begins again when finger is removed from screen
***Messages* window**	
Press, hold, and drag with one finger in Messages window.	Select all lines between where you first touch and then remove finger
Double tap one finger on *Messages* window Title Bar. (Docked or Floating)	Toggles the *Messages* window between Docked and Floating.
Double tap one finger on *Messages* window Title Bar. (Docked and Tabbed, during “fly-out”)	While the *Messages* window is “flown-out”, will change the *Messages* window from Docked and Tabbed (Retracted) to simply Docked.
Entity Editor, Meshing Toolbox, and PostProcessing Toolbox	
Tap one finger in any field or +/- toggle *Entity Editor*, *Meshing Toolbox*, or *PostProcessing Toolbox*.	Makes the field active for editing (except Read-Only fields). Tapping on the “Down arrow button” will bring up a drop-down menu and clicking on the “Action button” will bring up a dialog box. Also can be used to expand and collapse different sections by pressing the “+/-” toggle.
Tap one finger on title bar of an individual in *Meshing Toolbox* or *PostProcessing Toolbox*.	Makes the selected tool the “active” tool.
Press, hold, and drag with one finger in any field *Entity Editor*, *Meshing Toolbox*, or *PostProcessing Toolbox*.	Selects all characters in the field between where you press and release the mouse button.
Double tap one finger on window Title Bar. (Docked or Floating)	Toggles the *Dockable Pane* between Docked and Floating.
Double tap one finger on window Title Bar. (Docked and Tabbed, during “fly-out”)	While the *Dockable Pane* window is “flown-out”, will change the *Dockable Pane* window from Docked and Tabbed (Retracted) to simply Docked.
Data Table and Connection Editor	

Action	Result
Tap one finger on a row.	Selects and highlights the row the cursor is currently over. (Press Ctrl + the left mouse button to select multiple rows one at a time or Shift + the left mouse button to select a range of rows.)
Tap one finger on a column heading.	Toggles the Sort Method from lowest to highest (numerical and alphabetical) and vice versa.
Tap one finger on a column heading and drag the heading	Allows you to move the column headings around and place them in the order you wish. Also, dragging a column header off the row of column headers until an "X" appears over the column heading will remove the column from the *Data Table*.
Double tap one finger on window Title Bar. (Docked or Floating)	Toggles the *Dockable Pane* between Docked and Floating.
Double tap one finger on window Title Bar. (Docked and Tabbed, during "fly-out")	While the *Dockable Pane* is "flown-out", will change the *Dockable Pane* window from Docked and Tabbed (Retracted) to simply Docked
Model Info tree	
Tap one finger in *Model Info* tree on an entity.	Selects and highlights the entity in the tree the cursor is currently over. Also can be used to expand and collapse the different branches of the "tree", by pressing the "+/-" toggle and toggle visibility check boxes on/off.
Double tap one finger on *Model Info* window Title Bar. (Docked or Floating)	Toggles the *Model Info* tree between Docked and Floating.
Double tap one finger on *Model Info* window Title Bar. (Docked and Tabbed, during "fly-out")	While the *Model Info* tree window is "flown-out", will change the *Model Info* tree from Docked and Tabbed (Retracted) to simply Docked
Status Bar	
Tap one finger over an entity name in the "Tray" portion of the *Status Bar*.	Brings up a Context Sensitive Menu depending on the entity type the cursor is currently positioned over.

During a Command

When a command dialog box is displayed, touch can perform the following functions. These are also in addition to the normal Windows functions which allow you to move between dialog controls, and manage your windows. All picking must be done with a pointing device such as a mouse, styles, or "pen".

Action	Result
Graphics Window	
Press, hold, and drag with one finger in the Graphics Window	Dynamically rotates the model
Press, hold, and drag with two or more fingers together in the Graphics Window	Dynamically pans (translates) the model up, down, left, or right with minimal rotation of the model.
Press with two fingers apart from one another and move fingers towards each other (works with more than two fingers, but easiest with only two)	Zooms out.
Press with two fingers together and move fingers away each other (works with more than two fingers, but easiest with only two)	Zooms in
Press with two fingers and move one finger "around" the other finger (fingers apart from one another is easiest)	Rotates around the "Z-Axis of the screen" using the location of the stationary finger.

C Function Reference

This appendix defines the predefined FEMAP functions which can be used to create equations for any numeric input. The functions are listed in alphabetical order. All function arguments are always required. Multiple arguments must be separated by a semi-colon (;), not a comma (,).

ABS(x)

returns the absolute value of the argument x. If x was positive, ABS(x) = x.

ACOS(x)

returns the inverse cosine (arc cosine) of x. The inverse cosine is the angle, between 0 and 180 degrees, which has a cosine equal to x.

ACTID(type)

returns the active set or ID for the selected entity type. The argument, “type”, must be one of the predefined entity types (any other value of “type” returns an undefined result):

Type	Entity
1	Coordinate System
2	Point
3	Curve
4	Surface
5	Volume
6	Text
7	Boundary
8	Node
9	Element

Type	Entity
10	Material
11	Property
12	Load Set
13	Constraint Set
14	View
15	Output Set
16	Report Format
17	Connection
18	Connection Property

ASIN(x)

returns the inverse sine (arc sine) of x. The inverse sine is the angle, between -90 and 90 degrees, which has a sine equal to x.

ATAN(x)

returns the inverse tangent (arc tangent) of x. The inverse tangent is the angle, between 0 and 180 degrees, which has a tangent equal to x.

CNPR(cnpropID; index)

returns a property value from connection property “cnpropID”. The value of “index” determines the property value that will be returned. Refer to the table of Connection Property Values in Data Block 918 of the FEMAP neutral file documentation for proper values of “index”.

CNPRID(cnID)

returns the ID of the connection property that is referenced by connection “cnID”. The return will always be 0, if the connection does not reference a property.

COS(theta)

returns the cosine of the angle theta. Theta must be specified in degrees.

COUNT(type)

returns the number of entities of the selected type in the current model. Type must be one of the predefined entity types (see ACTID( ) function).

ELND(index;elemID)

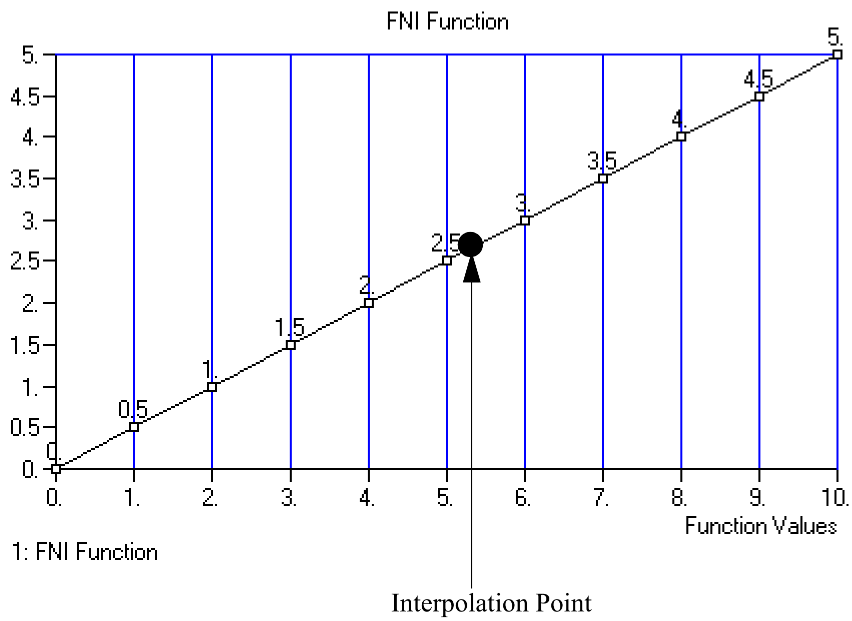
returns a nodeID which is referenced by an element. The first argument, index, selects which node on the element to report. The second argument, elemID, selects which element to report. For example ELND(3,45) returns the ID of the third node on element 45. If either the element does not exist, or the index is too large an error message will be given and the return will be undefined.

EXP(x)

returns the value of the exponential function, e^x.

FNI(functionID;x)

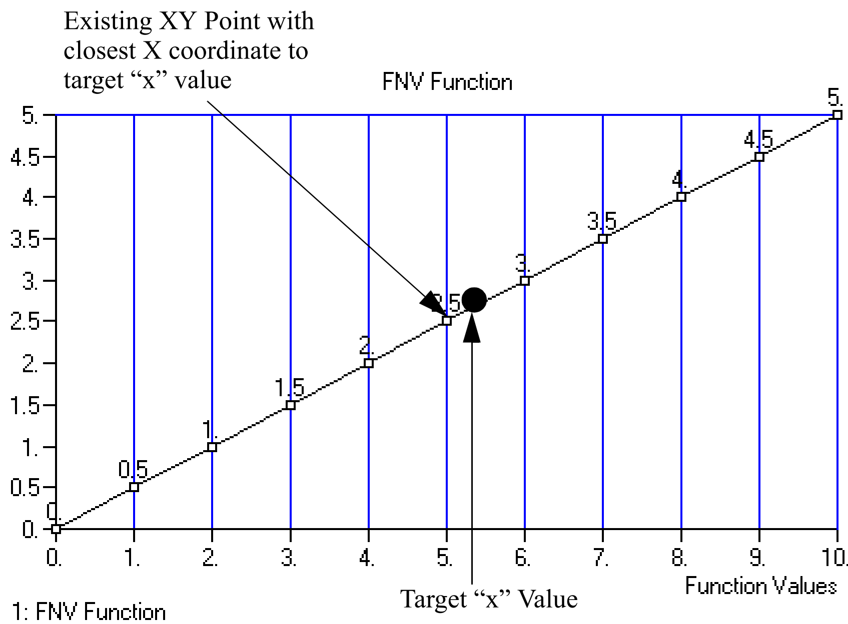
returns a Y value which has been linearly interpolated from existing data points in a user-defined FEMAP XY function. The type of function defined will have no effect on the values. For example, an XY function with the ID of "1..FNI Function" is defined using the Model, Function menu. The linear curve ranges from coordinates (0,0) to (10,5) with a data point a every integer X value (i.e. delta X is equal to "1"). A Y value is needed somewhere between two existing data points, for instance, at X=5.25. The function FNI(1;5.25) would return a value of 2.625. In addition, if an interpolation point is entered outside the range of the function (for the example, X values below 0 or above 10 would be out of the range), FEMAP will use either the first two or final two data points to determine the slope of the curve outside the range, and that curve only will be used to return Y values for out of range interpolation points.



FNV(functionID;x)

returns a Y value of the closest defined XY data point in a user-defined FEMAP XY function based on the target value of "x" in FNV(functionID;x). The target value "x" is used to determine which existing XY point has the closest X coordinate numerical value in the function. The type of function defined will have no effect on the values. For example, an XY function with the ID of "1..FNV Function" is defined using the Model, Function menu. The linear curve ranges from coordinates (0,0) to (10,5) with a data point a every integer X value (i.e. delta X is equal to "1"). A Y value is needed somewhere between two existing data points, for instance, at X=5.25. Since 5.25 is closer to 5 than it is to 6, the data point at X value 5 will be used, therefore the function FNI(1;5.25) would return a value of 2.5. In addition, if a target "x" value is entered outside the range of the function (for the example, X values below 0 or above 10 would be out of the range), FEMAP will use the start of end data point to determine the value of Y, and that Y value will re returned for all target "x" values outside the range of the function

.



INT(x)

returns the closest integer value (whole number) which is lower than the real number argument, x.

LN(x)

returns the natural logarithm of x.

LOG(x)

returns the base 10 logarithm of x.

MAT(matlID; index)

returns a material value from material “matlID”. The value of “index” determines the material value that will be returned. Unlike the PROP( ) function, the values of “index” in this case are the same, no matter what type of material is being referenced. You should use the following values for “index”.

Index Value	Returns	Index Value	Returns	Index Value	Returns
0	Ex	1	Ey	2	Ez
3	Gx	4	Gy	5	Gz
6	NUxy	7	NUyz	8	NUzx
9	G_3D[1,1]	10	G_3D[1,2]	11	G_3D[1,3]
12	G_3D[1,4]	13	G_3D[1,5]	14	G_3D[1,6]
15	G_3D[2,2]	16	G_3D[2,3]	17	G_3D[2,4]
18	G_3D[2,5]	19	G_3D[2,6]	20	G_3D[3,3]
21	G_3D[3,4]	22	G_3D[3,5]	23	G_3D[3,6]
24	G_3D[4,4]	25	G_3D[4,5]	26	G_3D[4,6]
27	G_3D[5,5]	28	G_3D[5,6]	29	G_3D[6,6]
30	G_2D[1,1]	31	G_2D[1,2]	32	G_2D[1,3]
33	G_2D[2,2]	34	G_2D[2,3]	35	G_2D[3,3]
36	alpha[1,1]	37	alpha[1,2]	38	alpha[1,3]
39	alpha[2,2]	40	alpha[2,3]	41	alpha[3,3]
42	k[1,1]	43	k[1,2]	44	k[1,3]
45	k[2,2]	46	k[2,3]	47	k[3,3]

Index Value	Returns	Index Value	Returns	Index Value	Returns
48	thermal cap	49	density	50	damping
51	ref Temp	52	tension limit[1]	53	tension limit[2]
54	compression limit[1]	55	compression limit[2]	56	shear limit

MAX(x;y)

returns either x or y, whichever is larger. Positive numbers are always larger than negative numbers. If you want to compare in an absolute sense use MAX(ABS(x);ABS(y)).

MAXID(type)

returns the maximum ID in the current model of the selected entity type. Type must be one of the predefined entity types (see ACTID( ) function).

MID(propID)

returns the ID of the material that is referenced by property "propID". This function should not be used with laminate properties which can reference multiple materials - use MLAM( ) instead.

MIN(x;y)

returns either x or y, whichever is smaller. Negative numbers are always smaller than positive numbers. If you want to compare in an absolute sense use MIN(ABS(x);ABS(y)).

MINID(type)

returns the minimum ID in the current model of the selected entity type. Type must be one of the predefined entity types (see ACTID( ) function).

MLAM(propID; ply)

returns the ID of the material that is referenced by layer "ply" of property "propID". This function can only be used if "propID" selects a laminate property. "ply" must be between 1 and the maximum number of allowable plies for a laminate, however the return will be 0 if you choose a ply that is not defined for the selected property.

NEXTID(type)

returns the ID of the next entity to be created of the selected type. Type must be one of the predefined entity types (see ACTID( ) function).

PID(elemID)

returns the ID of the property that is referenced by element "elemID". The return will always be 0, if the element does not reference a property.

POW(x; y)

returns the value of x to the y power, x^y

PROP(propID; index)

returns a property value from property "propID". The value of "index" determines the property value that will be returned. Refer to the table of Property Values in Data Block 402 of the FEMAP neutral file documentation for proper values of "index". As you can see from this table, the "index" values differ depending on the type of property that you are using.

For example, if propID selects a Bar property, setting index = 0 will return Area, index = 4 will return J, the torsional constant. For a plate, index = 0 will return the thickness.

RND(x)

returns the closest integer value (whole number) which is either lower or higher than the real number argument, x.

SIN(theta)

returns the sine of the angle theta. Theta must be specified in degrees.

SQR(x)

returns the square of x. SQR(x) = x * x

SQRT(x)

returns the square root of x.

Note: Please be very careful when using the SQR and SQRT functions in different portions of FEMAP. When working within the FEMAP interface, such as creating an equation for loading, SQR is “square”, while SQRT returns the “square root”. When creating a “script” using the *API Programming* window (see Section 7.2.10, "Tools, Programming, API Programming"), SQR will actually return the “square root”, not “square” the value.

TAN(theta)

returns the tangent of the angle theta. Theta must be specified in degrees.

VEC(setID;vectorID;entityID)

returns an output data value. SetID defines the output set to be selected. VectorID selects an output vector in that set. EntityID is either the element ID or node ID (depending on the vector type) of the data to be selected. For example, VEC(2,1,33) returns the output value for node 33, in Output Set 2, Output Vector 1 (Total Translation).

XEF(elemID;faceID)

returns the X coordinate of the centroid of the selected element face. The X coordinate is always returned in the active coordinate system. In a cylindrical or spherical system, this is the radial value. The available values for faceID depend on the element type. Refer to the Element Library for more information on face numbers for each element type.

XEL(elemID)

returns the X coordinate of the centroid of the selected element, in the active coordinate system. In a cylindrical or spherical system this is the radial value.

XND(nodeID)

returns the X coordinate of the selected node, in the active coordinate system. In a cylindrical or spherical system this is the radial value. If you specify a negative nodeID, FEMAP selects the node with an ID equal to the next node to be created minus the value you specified. For example, if you specify XND(-1), and the next node to be created is 43, you will get the X coordinate of node 42, if node 42 exists.

XPT(pointID)

same as XND( ), only returns the coordinates of a point.

YEF(elemID;faceID)

same as XEF( ), only returns the Y coordinate.

YEL(elemID)

same as XEL( ), only returns the Y coordinate.

YND(nodeID)

returns the Y coordinate of the selected node, in the active coordinate system. In a cylindrical system this is the angular value theta, in degrees. See XND( ) for additional information.

YPT(pointID)

same as YND( ), only returns the coordinates of a point.

ZEF(elemID;faceID)

same as XEF( ), only returns the Z coordinate.

ZEL(elemID)

same as XEL( ), only returns the Z coordinate.

ZND(nodeID)

returns the Z coordinate of the selected node, in the active coordinate system. See XND( ) for additional information.

ZPT(pointID)

same as ZND( ), only returns the coordinates of a point.

Converting Old Models

D

Conversion from v4.4 and Later

FEMAP binary database (.MOD) files from previous versions of FEMAP are not compatible with this version. The current version of FEMAP, however, can automatically translate model files from FEMAP v4.4 and later to the current version. No action is required by the user. FEMAP will automatically read the database, translate to a FEMAP neutral file, and then convert the neutral file to the latest database format.

Conversion of v4 files - Prior to v4.4

If you have been using a version of FEMAP prior to v4.4, you will probably have existing models that you want to use with this new version. Your old model (.MOD) files are not compatible with this version of FEMAP and cannot be read by the latest version of FEMAP. You can convert your models to FEMAP neutral files, however, so they can be used with this system.

The best approach to doing the conversion is to do the following:

1. Using your old version of FEMAP, open the model that you want to convert.

2. Use the *File, Translate* command (pre Version 5), or the *File, Export, FEMAP Neutral* command, to write a FEMAP neutral file. You can choose any filename for this command. If your model contains output for post-processing you can also write that to the neutral file for automatic conversion.

3. Exit your old version of FEMAP.

4. Start the new version of FEMAP with a new, empty model.

5. Choose the *File, Import, FEMAP Neutral* command.

6. When the list of files is displayed, enter the name of the neutral file that you just created. You can set any of the other translation options that you want, but usually you will just want to press *OK* and accept the defaults. FEMAP will now read the neutral file and build your converted model.

7. Before you exit FEMAP, remember to save your model. Either use the *File, Save As* command or just respond *Yes* to the question about saving your model as you exit FEMAP.

Other Methods to Convert Models

The above approach used a FEMAP neutral file to convert your model. You can follow the same steps using any of the other translator formats that are supported in your earlier version of FEMAP. If you use one of these alternate formats however, no colors, groups, or other FEMAP-specific data will be converted.

Converting from Version 3 and Before

If you still have the output files from your analysis program, it will probably be better to skip writing the output to the neutral file. Due to the large number of changes in this version of the program, output that is translated from neutral files that were written by earlier versions of FEMAP will be moved to output vectors that it would not otherwise occupy. You will still be able to post-process this output, but if you later read additional analysis output into new output sets, it will not be in the same vectors as the output converted via the neutral file. For maximum compatibility, it is therefore best to skip writing output to the neutral file and reread your analysis output directly into this version of FEMAP.

If you do this, you will have to use *File, Import, Analysis Results* to read the output files into the newly converted model.

Due to the extensive enhancements that were made between Version 3 and 4 of FEMAP, significant changes had to be made to the neutral file format. Therefore, even though you follow the procedure defined above, your complete model cannot be converted exactly. Again, these limitations only apply if you are converting a neutral file from Version 3 or before - not a Version 4 neutral file. The following items will either not be converted, or will be modified during the conversion:

- As noted above, all output will be moved to new output vectors. The new vectors will all have IDs above 300000, just like user-defined output.
- Display, Window, and Post options will be skipped. The new views, which combine all of these features and many more, are so different that it is impossible to convert these old options. You must define new views.
- PostProcessing report formats will be skipped. Again, these have changed so much that they cannot be converted. Either select new standard formats or redefine them in your new model.
- Groups will be converted, but the format has changed dramatically. Some types of group definition, like constraints by DOF, cannot be converted because they are no longer supported. If you see any messages when reading the neutral file, you should check your groups carefully.

Index

Q

R

S

T

U

V

W

X

Y

Z