What's New

Femap 2022.1

About Siemens Digital Industries Software

Siemens Digital Industries Software is a leading global provider of product life cycle management (PLM) software and services with 7 million licensed seats and 71,000 customers worldwide. Headquartered in Plano, Texas, Siemens Digital Industries Software works collaboratively with companies to deliver open solutions that help them turn more ideas into successful products. For more information on Siemens Digital Industries Software products and services, visit **www.siemens.com/plm**.

Support Center: **support.sw.siemens.com**

Send Feedback on Documentation: **support.sw.siemens.com/doc_feedback_form**

Contents

1. Femap What's New

2. What's New - version 2022.1

The following chapters describe new features, changes and enhancements to Femap for version 2022.1:

User Interface

General

- Added a **Add to Copied Entity Groups** option to the **Parameters** section in the Copy Option dialog box for **Geometry → Copy/Rotate/Reflect** commands for all geometry types.

- Added the **Nonstructural Mass Property Values** section to the Define Property - BEAM Element Type dialog box (**Model → Property** – *Beam* command). Renamed the **Nonstruct Mass/Length** option to **Nonstructural Mass/Length**, and moved it from **Property Values** to the new **Nonstructural Mass Property Values** section.

- Changed the name of the **Autodesk Nastran** tab to **Simcenter/Autodesk Nastran** in the **Model → Property** command for Plate element Property Extensions dialog box.

Menu

- Added the **HexMesh Bodies...** command to the **Mesh** menu. See **Hex-Dominant Mesher** for more information.

- Added the **Mapped Hex Refine...** command to the **Mesh → Editiing** submenu. See **Hex-Dominant Mesher** for more information.

- Added the **Kinematic Driver...** command to the **Model → Load** submenu. See **Kinematic Joints** for more information.

- Added the **Kinematic Joints...** command to the **Model → Simulation Entities** submenu. See **Kinematic Joints** for more information.

- Added **Kinematic Joint...** to the **Delete → Model** submenu.

- Added **Kinematic Joint** to the **Group** menu. The **Kinematic Joint...** item contains submenu items **ID**, **Color** and **Layer**.

- Added the new Node\Element Entity Eelection methods to the **Group → Node** and **Group → Element** submenus. See **UI/Usability Features** for more information.

- Added **Flexible Sliders** to the **Model → Simulation Entities** submenu. See **Flexible Sliders** for more information.

- Added **Flexible Slider...** to the **Delete → Model** submenu.

Model Info Tree

- Added the ability to **Evaluate** Groups from within the **Model Info** tree. See **UI/Usability Features** for more information.

- Added **Kinematic Joints** to the **Simulation Entities** branch of the **Model Info** tree. See **Kinematic Joints** for more information.

- Added **Flexible Sliders** to the **Simulation Entities** branch of the **Model Info** tree. See **Flexible Sliders** for more information.

Entity Selection

- Added Methods to select nodes and elements on points, curves, and surfaces of solids. See **UI/Usability Features** for more information.

- Added Methods to select nodes and elements on points, curves, and surfaces of solids. See **UI/Usability Features** for more information.

Preferences

- **Interfaces** tab - Added new settings in the **Nastran Options** section; **Activate OGEOM Parameter**, **Results Dest.** (**Femap** or **Simcenter**) to assist requesting Nastran analysis output to the *.op2* file. See **Request Output To .op2 File** for more information.

- **Color** tab - Added a **Kinematic Joint** item to **Entity Colors**. See **Kinematic Joints** for more information.

- **User Interface** tab - Added the **Help...** button. Opens the Help Options dialog which is used to select the Femap help source (Online HTML or Local HTML - requires Siemens PLM Documentation Server). See **New Help System** for more information.

Hex-Dominant Mesher

Hex-dominant meshing of solid geometry with little to no simplification or subdivision into smaller and simpler regions has long been desired by the finite element analysis community. Simcenter Femap has collaborated with other development organizations in the Siemens Simcenter Portfolio in order to offer this exciting technology to Femap users for the first time for v2022.1.

To accomplish this goal, the hex-dominant mesher first fills a solid volume with as many hexahedral elements as possible, then fills the remainder of the volume with wedge, pyramid, and tetrahedral elements, as needed. This process creates high quality elements which can be sent directly to the

Simcenter Nastran solver with no additional interaction from the user or manual refinement of the mesh.

In addition to being able to mesh single parts, the hex-dominant mesher can work on multiple parts to create a single continuous mesh of the assembly. There are also additional options to control various aspects of mesh sizing, mesh associativity, and if all elements should be given midside nodes during the meshing process.

The following two commands are added for Hex-Dominant Hexahedral Meshing.

Automatic Hex-Dominant Meshing of CAD Geometry

Previous versions of Femap provide semi-automatic , but it requires you to split the geometry into sections so Femap could setup connected mesh sizing and sweepable areas. This can be time consuming, and is essentially an intuitive process.

Added a **HexMesh Bodies...** command to the **Mesh** menu.
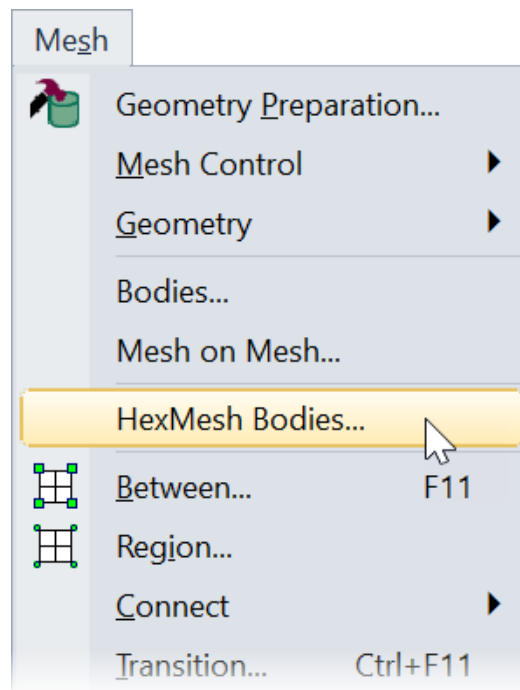
The new **HexMesh Bodies...** command eliminates the need to split the geometry. Selecting the **Mesh → HexMesh Bodies...** command will open the Hex-Dominant Automesh Bodies dialog box after the geometric entities are selected.
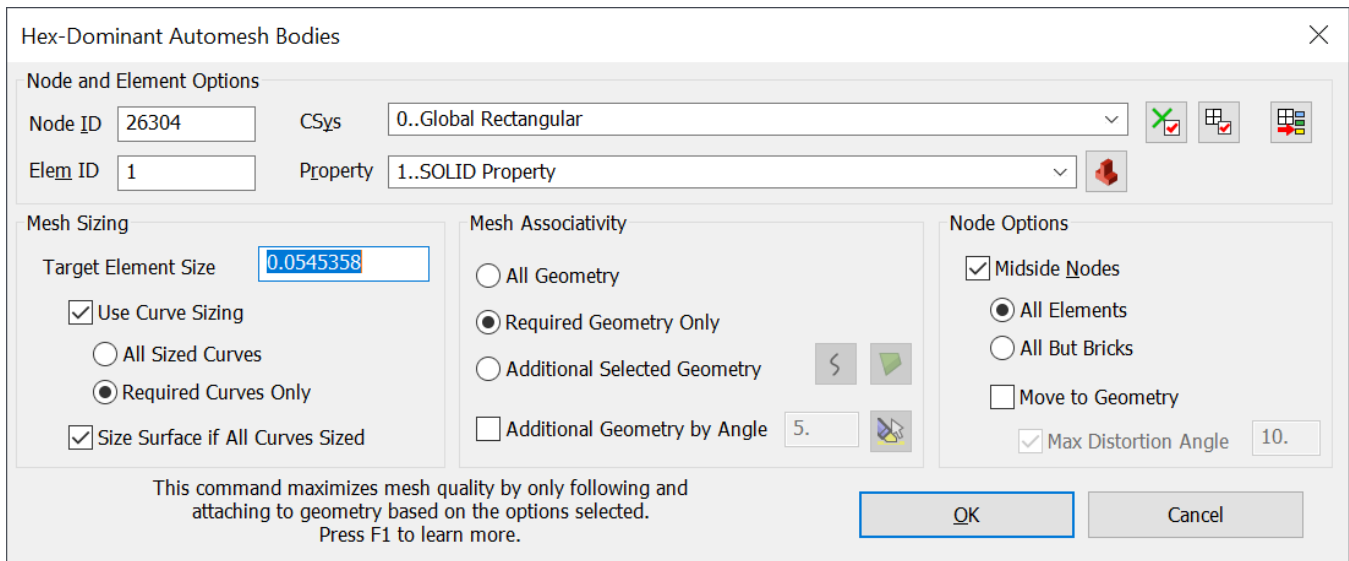
Constrained nodal locations make it difficult to get an acceptable hex-dominant mesh, so options are therefore very similar to the **Mesh → Bodies...** command. You are given a **Target Element Size**, not an exact one.

The **Mesh Associativity** section is where required geometry is associated, or geometry with Loads, Boundary Conditions, Contact Regions, etc. You can also request associativity on additional entities if required.

Node Options control how the **Midside Nodes** are handled, or the element order and projection to geometry.

The mesher can take some time to process. When the meshing is finished and the **Midside Nodes** are processed, you end up with a Hex-Dominant Mesh.
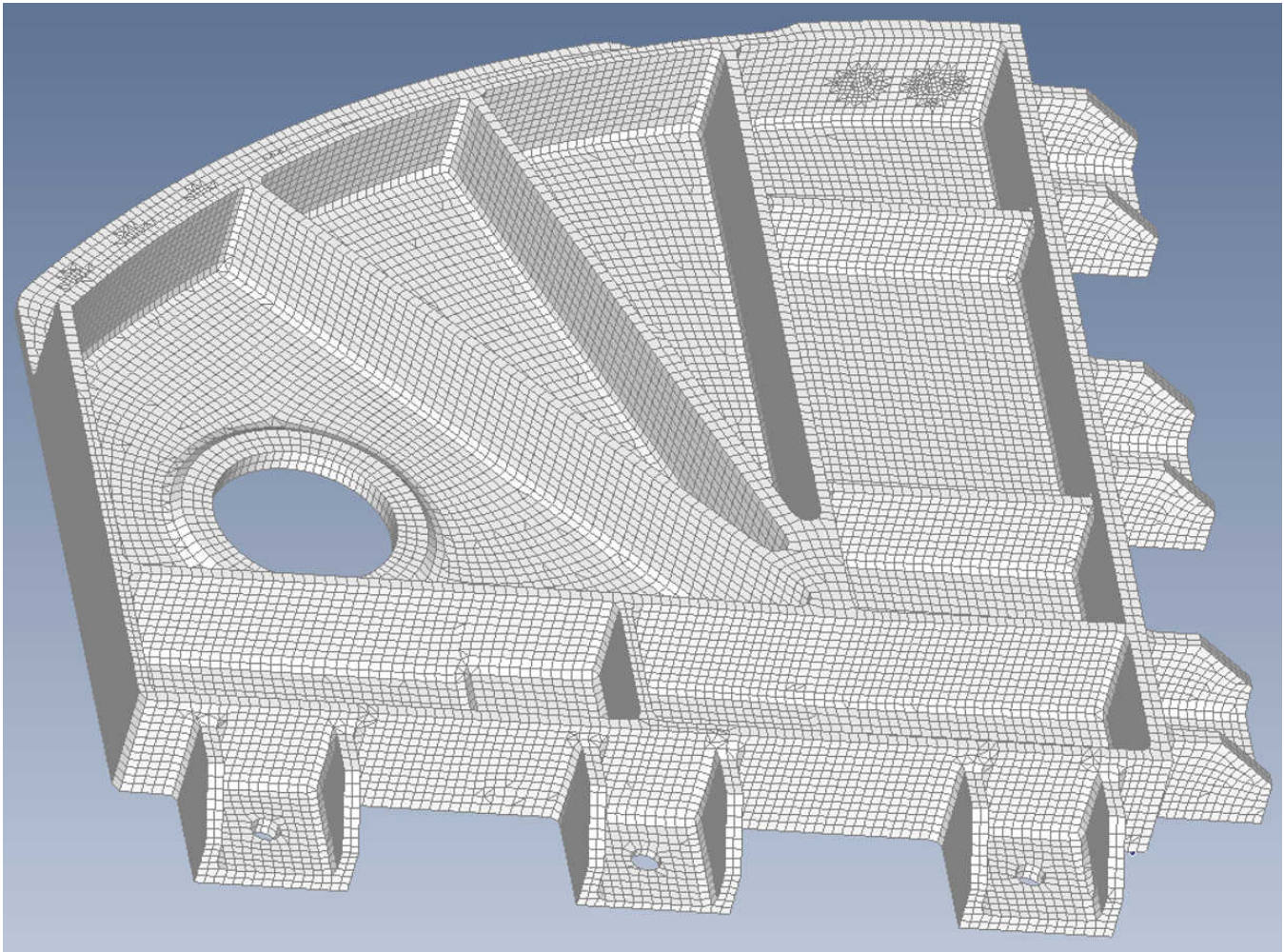
will then display statistics in the **Messages** window. Counts of Hex, Wedge, Pyramid and Tet elements are listed. Also, listed is the percentage of Hex elements by volume. In the example here, 96% of the volume is encompassed by hexihedral elements.
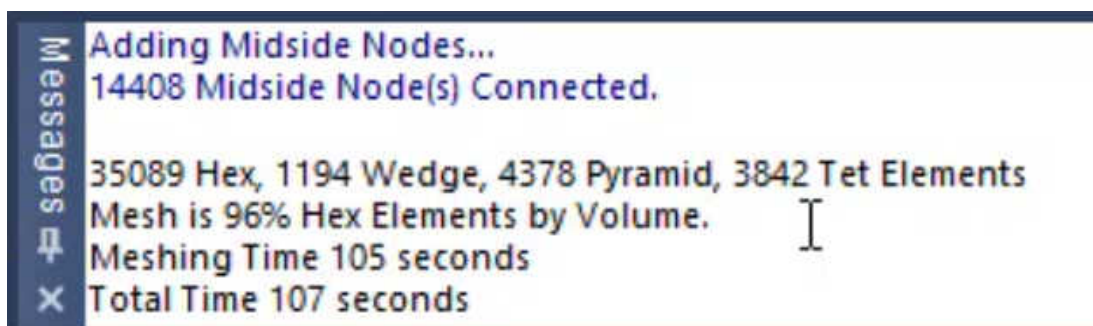
Use the **Model Tree** to view the mesh elements (**Model → Elements → By Shape**):
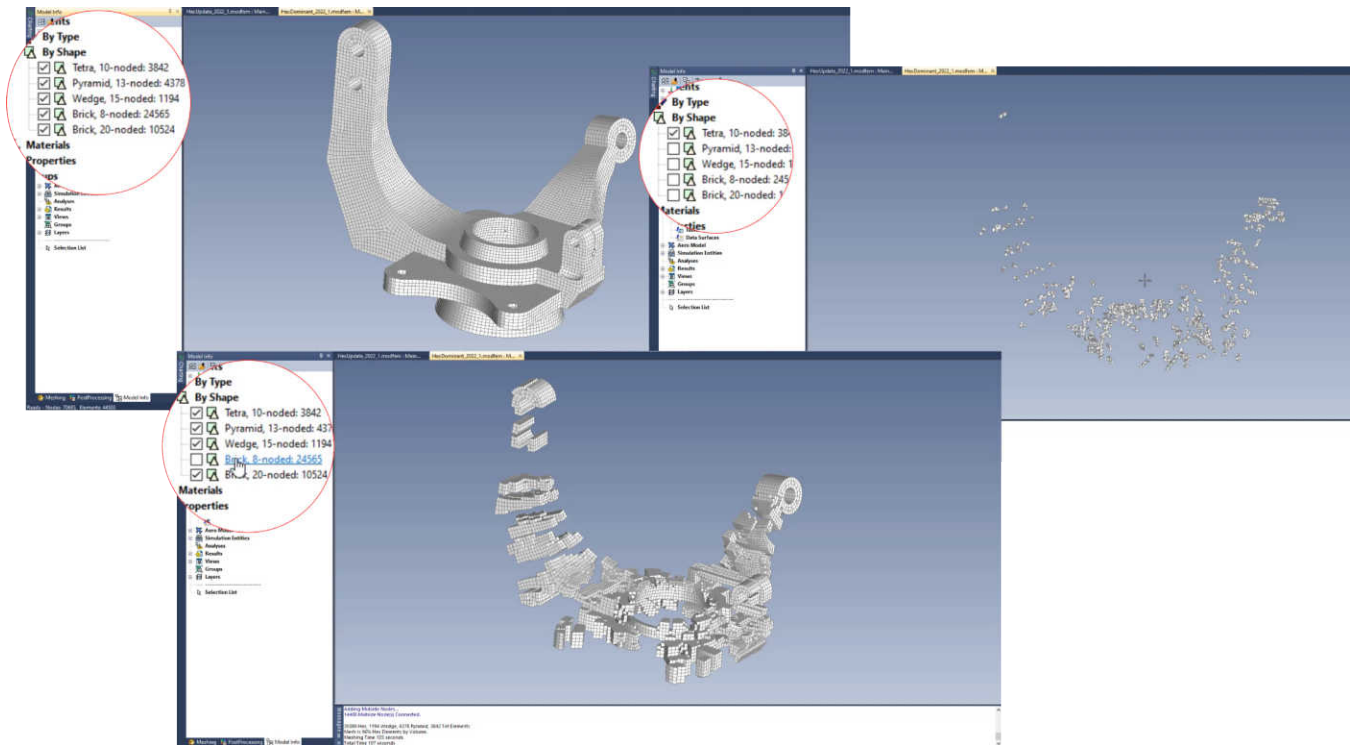
Using the **Check Element Quality** command (**Tools → Check → Element Quality...**) will indicate the elements pass the Nastran checks.
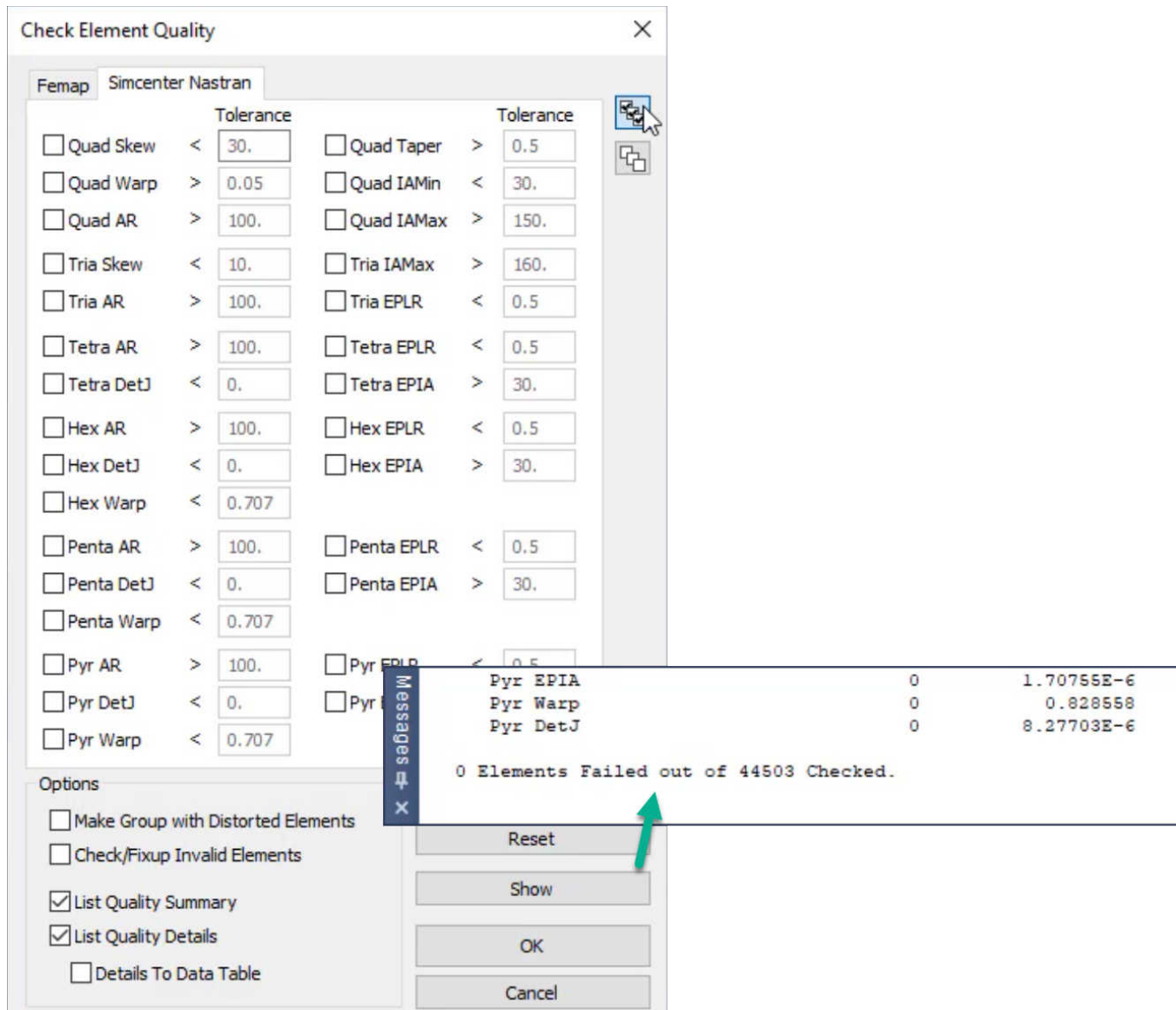
Automatically Refine Hexihedral Meshes

Added a **Mapped Hex Refine...** command to the **Mesh → Editiing** submenu.
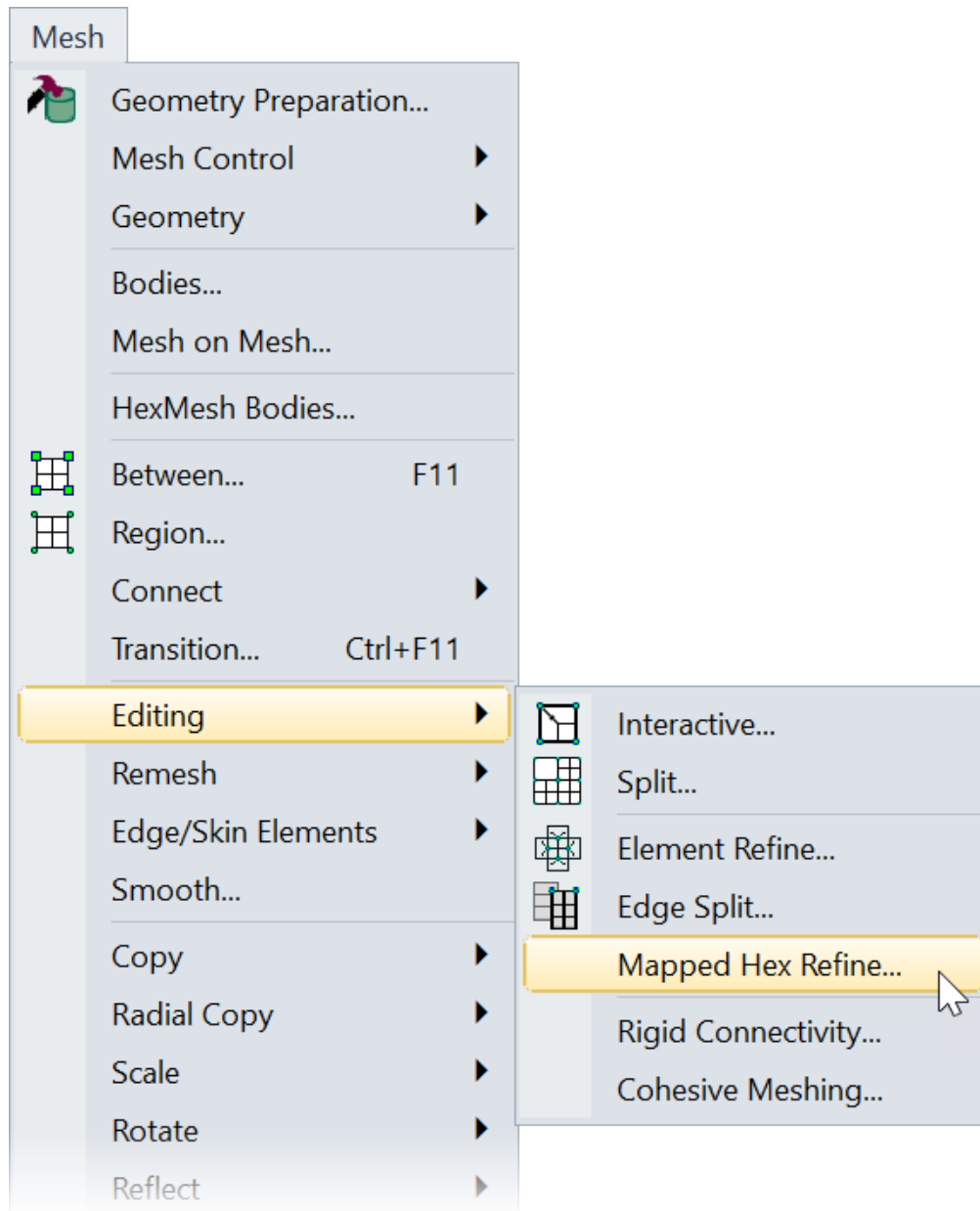
Selecting **Mesh → Editiing → Mapped Hex Refine...** opens the Mapped Hex Refinement dialog box for the selected mesh.
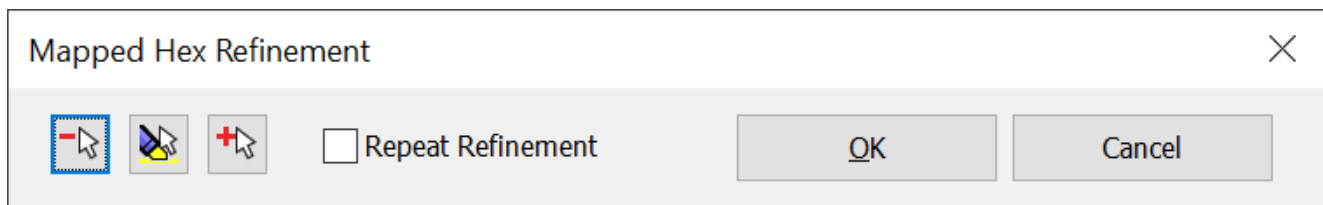
Pick the hexahedral elements you want to refine, and click OK. The command splits all the hex elements, and produces a more refined hex mesh in the local area. Checking **Repeat Refinement** automatically refines down another layer.
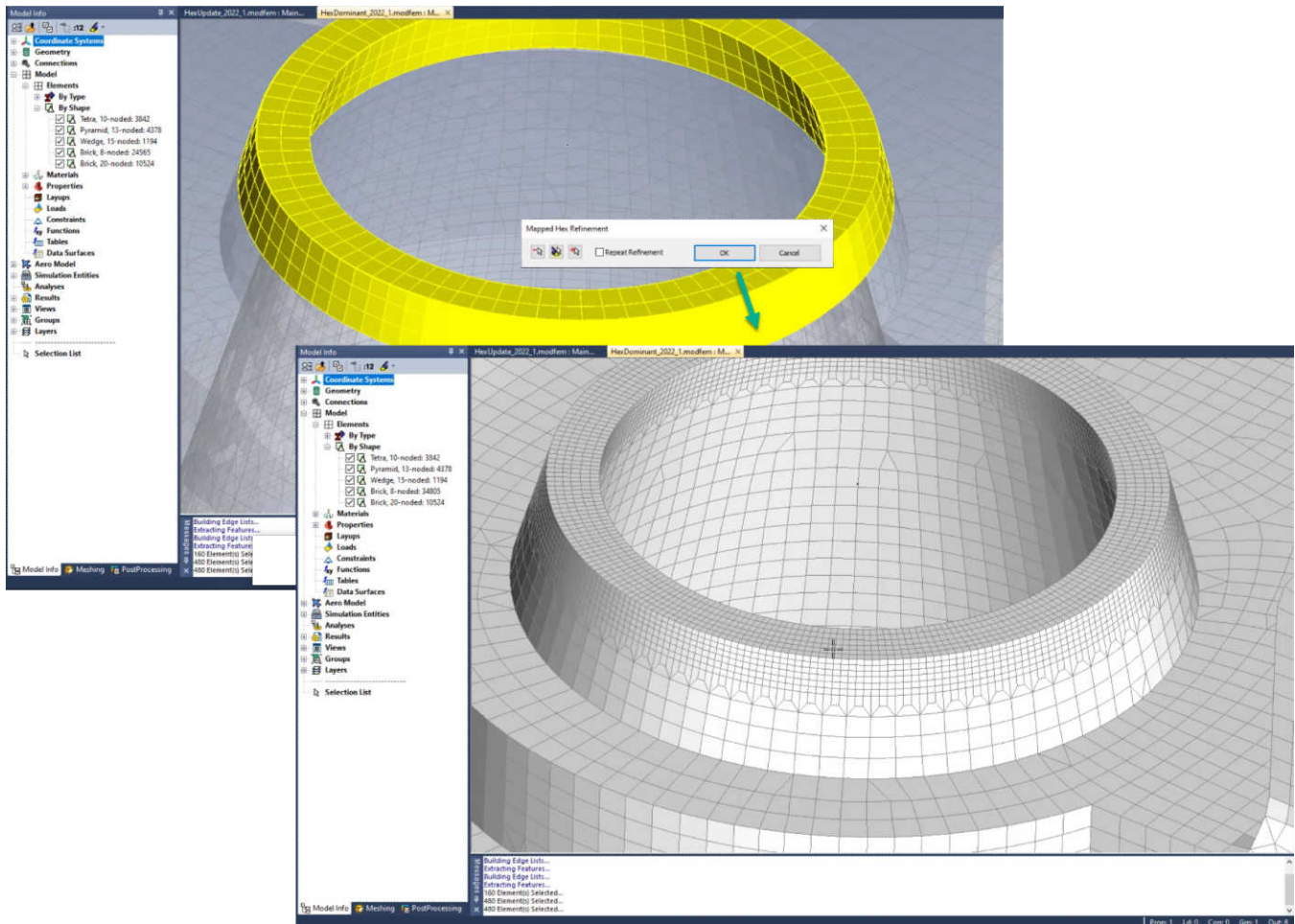
Note:

The **Mapped Hex Refine...** command is used for traditional Mapped Hexahedral Meshes, not mixed Hex-Dominant Meshes.

Kinematic Joints

Simcenter Femap will begin to provide support for advanced simulation methods, such as flexible body dynamics, using Simcenter Nastran's Multi-Step Nonlinear Kinematic solution, SOL 402. As a first step, Kinematic Joints have been added as Simulation Entities for version 2022.1.

Kinematic Joints allow for structural analyses of assembies containing moving parts. Kinematic Joints connect two nodes and allow a certain relationship of relative motion or rotation between the two depending on what type of joint is established. The types of Kinematic joints include revolute, inline, slider, spherical, cylindrical, and other highly specialized types which can be used for the analysis of
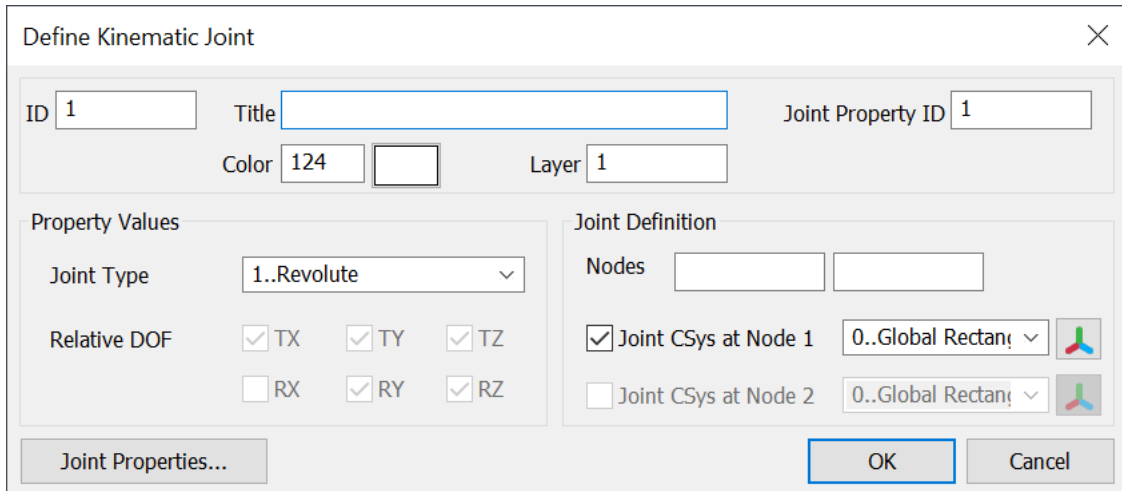
models containing moving parts, such as aerostructures, helicopters, deployable structures in space, gas turbines, and machine tools. The addition of drive loads as a new load type can be used to enforce displacements or rotations on kinematic joints.

To accelerate the process of creating kinematic joints, Kinematic Connections have also been added, which are a type of entity unique to Femap. Kinematic Connections allow the user to establish how a kinematic joint will be connected to geometry or existing mesh of the model and are then expanded to the necessary nodes when Simcenter Nastran input file is exported.

Kinematic joints are found under the **Simulation Entities** branch in the **Model Info** tree.

Defining Kinematic Joints

Select the Kinematic Joints element from the **Model Info** tree, right click, and choose 🔧 **Kinematic Joint** from the menu - Or select from the main menu: **Model → Simulation Entities →** 🔧 **Kinematic Joints...**. This opens the Define Kinematic Joint dialog box.

You can create various joint types by selecting from the **Joint Type** drop-down. The options are **1..Revolute, 2..Inline, 3..Slider, 4..Spherical, 5..Cylindrical, 6..Slider-Universal, 7..Fixed, 8..Screw, 9..Universal** and **10..Constant Velocity**.

The example below, the model has four revolute joints connecting the suspension linkage on the rear of a mountain bike. This allows you to connect the different pieces of the assembly, and also allows them to rotate.
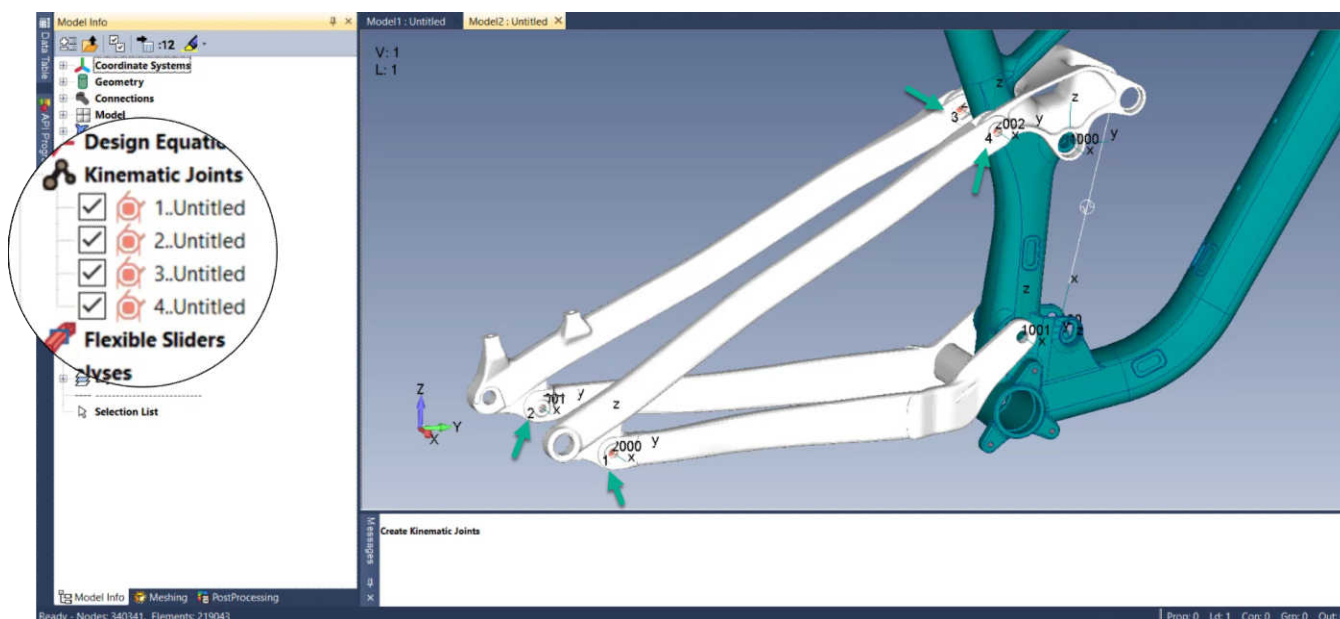
With this model solved, it can be animated to show the deformation of the rear suspension travel.

As the frame articulates, the revolute joints connect the parts of the geometry in the assembly, and allow the pieces to rotate about the point of the revolute joint.

Defining Kinematic Joint Connections

There has also been the addition of kinematic connections. These kinematic connections speed up the process of creating kinematic joints.

Select the Kinematic Joints branch from the **Model Info** tree, right click, and choose 🔗 **Kinematic Connection** from the menu. This opens the Define Joint Connection dialog box.

For instance, to create a revolute joint on one side of some universal joint geometry, set it **To Ground** because it is attached to nothing on the open end. Then connect the motion attachment as a surface.

Femap automatically generates a coordinate system in the axis that is needed, along with the point of the joint location.
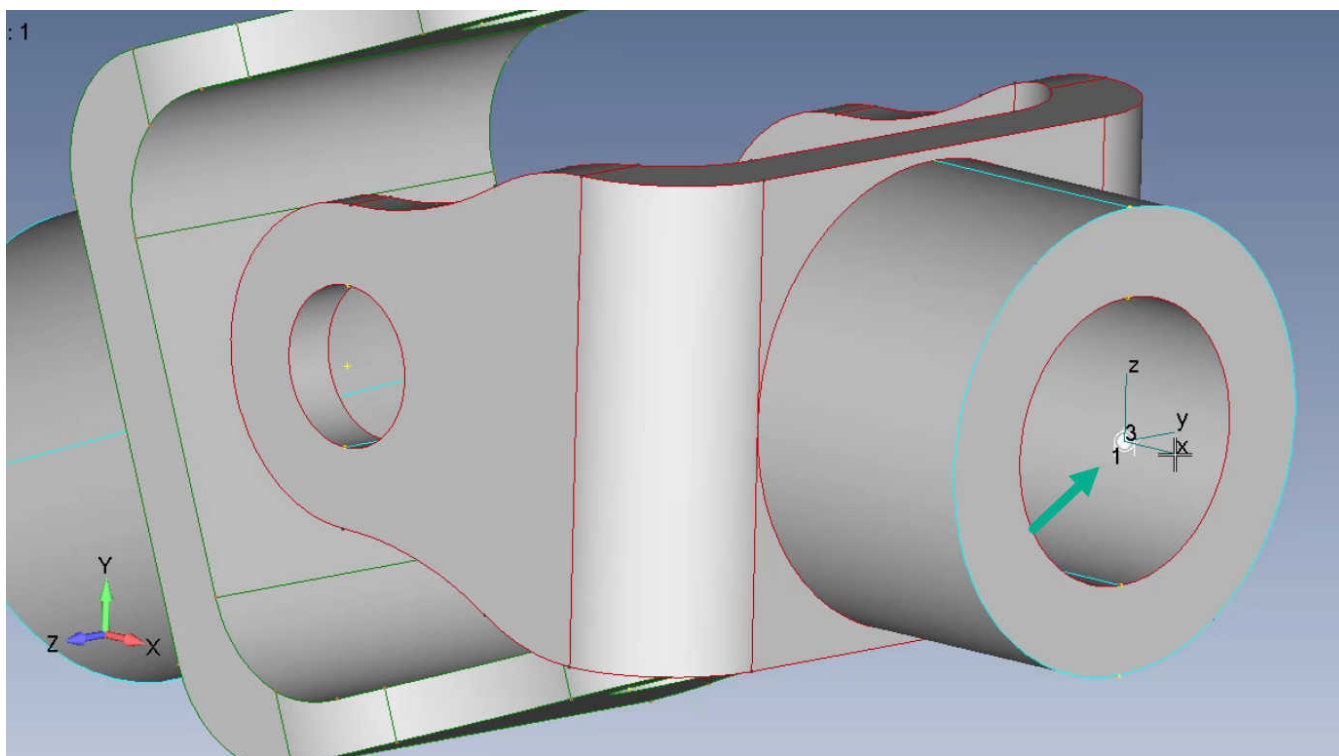
Click **Joint Properties...** (opens the Joint Properties dialog box) to specify some of the more technical things like translation stiffness - **Translation (KT)**, rotational stiffness - **Rotational (KR)** and others depending on the type of joint.
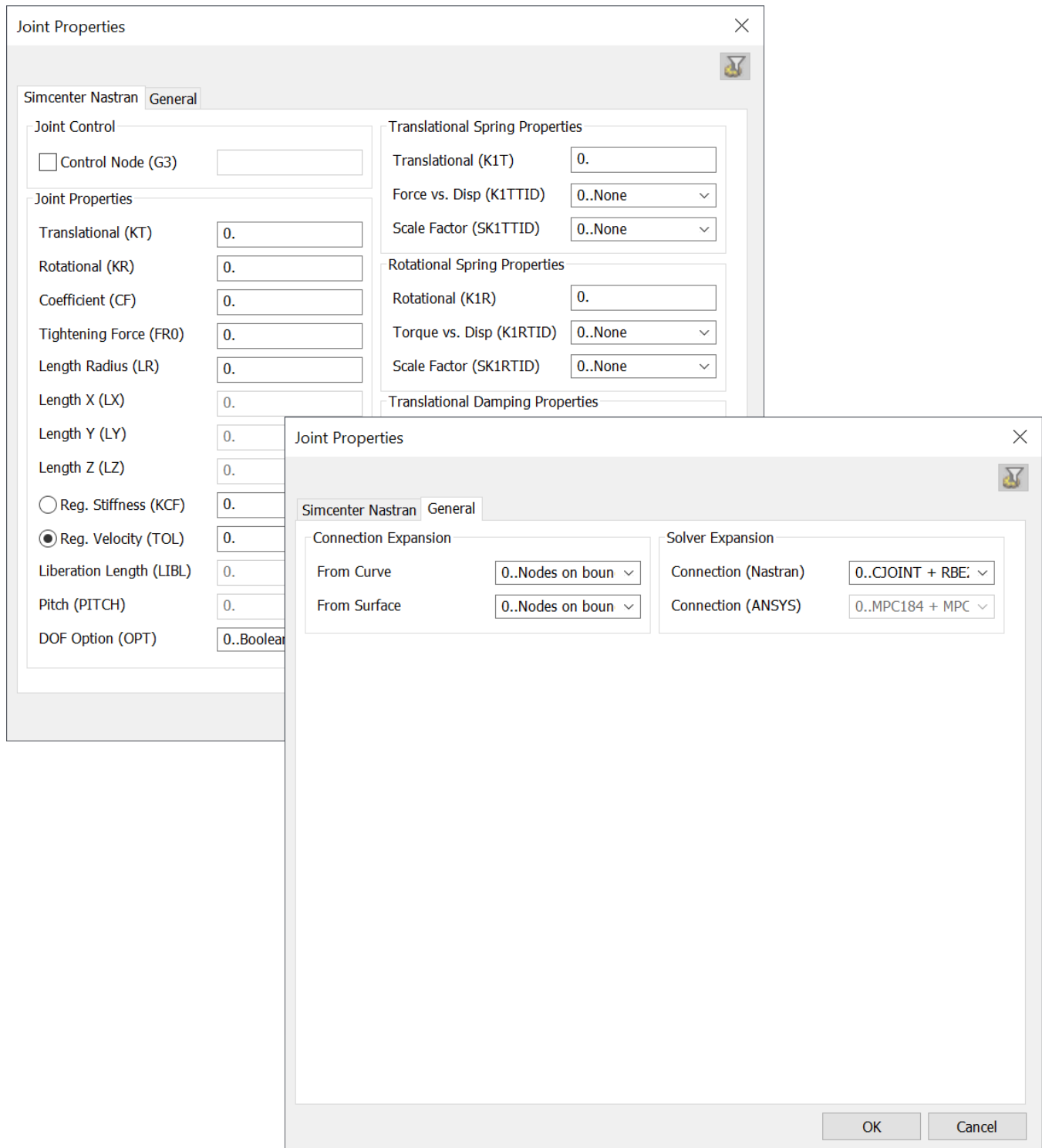
On the **General** tab of the Joint Properties dialog box, you can set some of the **Connection Expansion** methods. For instance, select a surface to connect to so you can choose whether it expands out to **0..Nodes on boundary only**, **1..Nodes in interior only**, or **2.. Nodes on boundary and interior**.

For the Nastran expansion - **Connection (Nastran)** - you select what type of element will connect the joint to the mesh.

With more kinematic connections added creating a universal joint, the motion can be simulated. First, the surface of the holes (**1**) that make up the clevis and a coordinate system with the x-axis (**2**) oriented to align with the holes. Secondly and similarly, on the other side of the assembly, the holes (**3**) are aligned to the other coordinate system (**4**). This completes the universal joint.
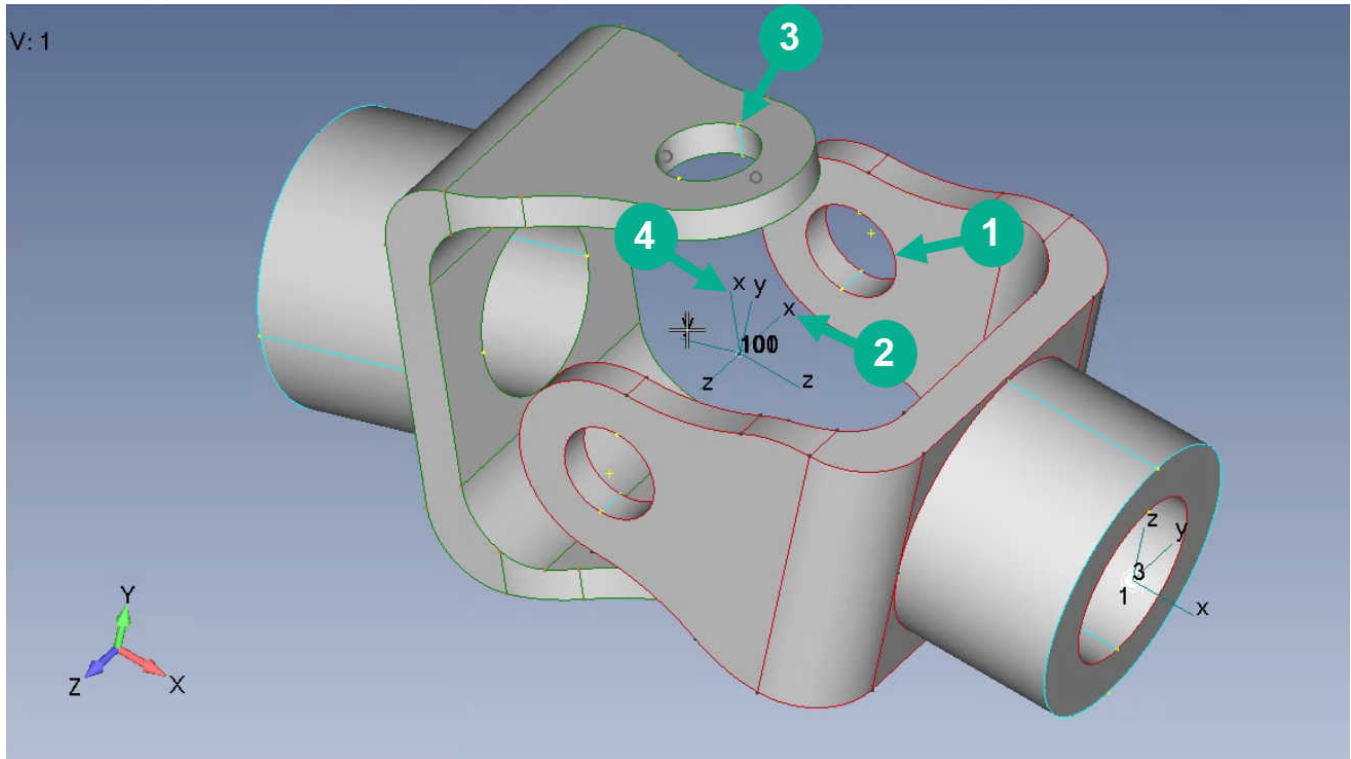
Create Loads on Kinematic Joints

To drive the motion of the simulation, a driver load is needed on the revolute joint. The new **Kinematic Driver...** command is used to accomplish this. Select **Model → Load → Kinematic Driver...** to start the command. The menu selection opens the Create Loads on Kinematic Joints dialog box.
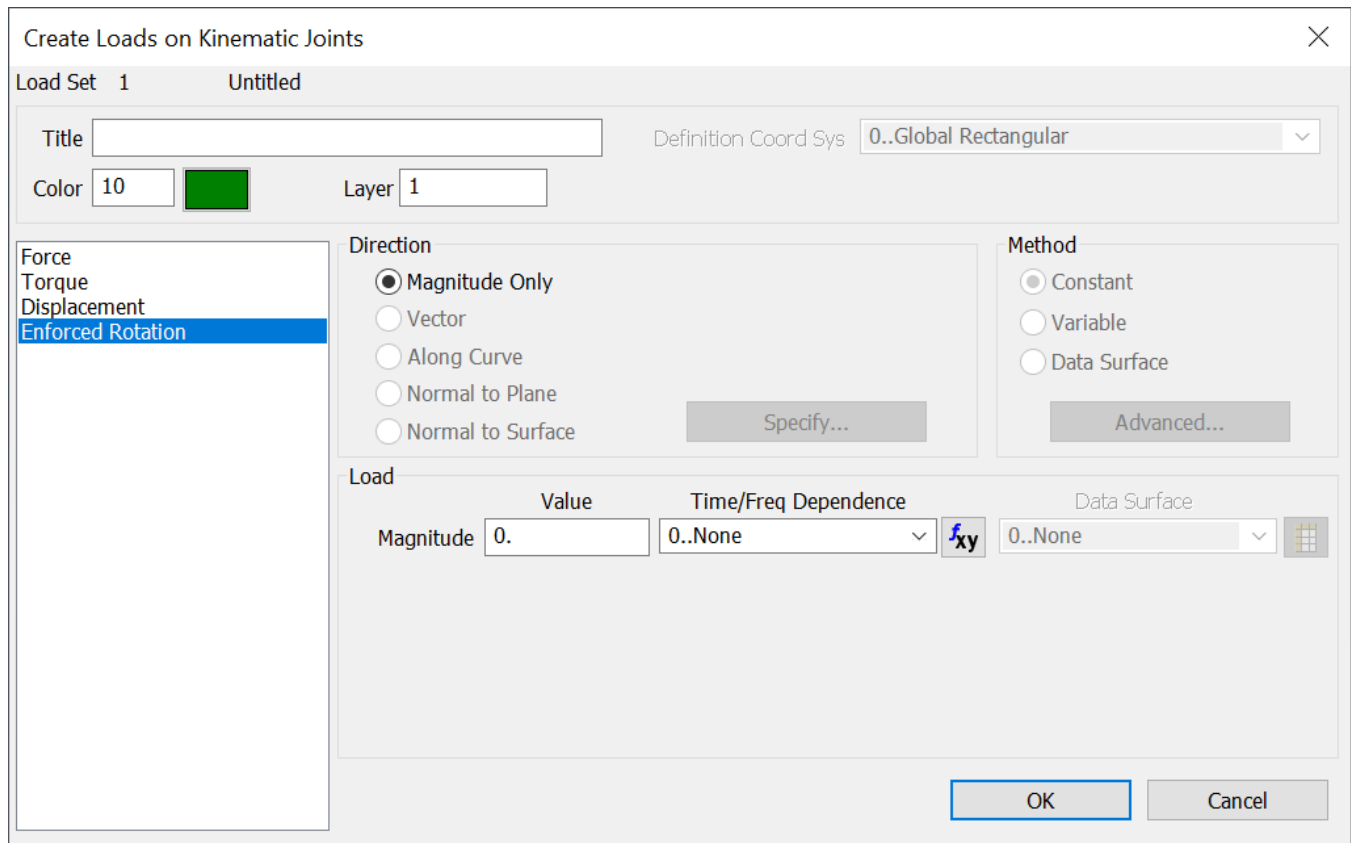
Select the **Enforced Rotation** load, and enter a **Magnitude** of 360 degrees (in radians) and apply it to the joint (click OK).

Managing Kinematic Joints

The right click menu in the Kinematic Joints branch of the **Model Info** tree has several additional options.

> Note:
>
> The menu is available if you click on the branch and the individual kenematic joints.

The **Kinematic Joint Manager** menu option opens Kinematic Joint Manager dialog box.
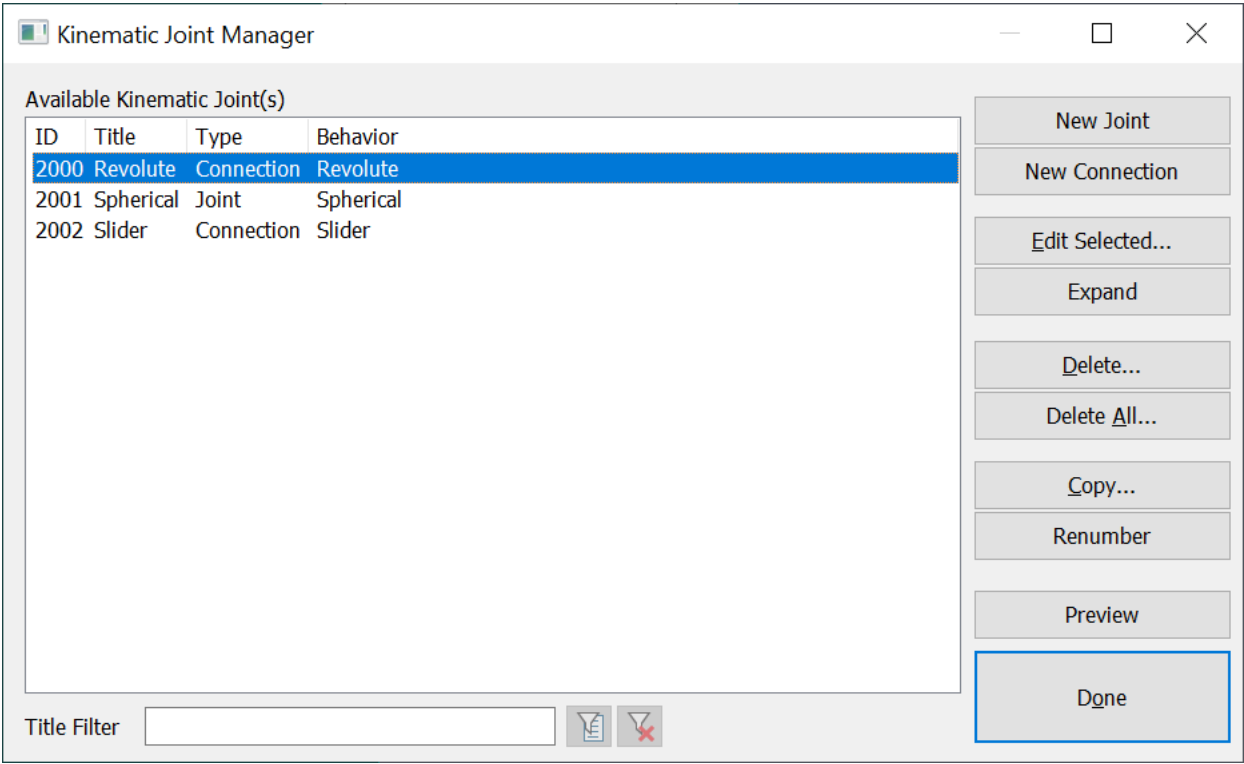
Here, the Kinematic Joints in the model are listed. You can do some of the same actions mentioned previously such as creating a **New Joint**, or a **New Connection**. Edit kinematic joints by clicking **Edit Selected...** (opens the Define Joint Connection dialog box). You can delete the selected kinematic joint (click **Delete...**) or delete all the kinematic joints in the model (click **Delete All...**). There are also options to **Expand**, **Copy...**, **Renumber** and **Preview** the kinematic joints in the model. Finally, if you have a long list of kinematic joints, you have the option to filter by title (enter text into the **Title Filter** field).

The **List** menu option lists information on all the kinematic joints in the **Messages** pane. For example:

```
              ID
Title
Type                 Node 1      Node 2      CSys 1      CSys2
_____
_____
_____
              2001
Spherical
         Spherical         2301        2302         0          0


              ID
Title
Type                 Attach      Attach 2    CSys 1      CSys2
_____
_____
```

```
_____
          2000
Revolute
          Revolute          Surfaces   Nodes      0           0
          2002
Slider
          Slider            Surfaces   Surfaces   0           0
```

The **Color** and **Layer** options are used to manipulate the kinematic joints display and layer assignment. They open the Color Pallete and Select Layer dialog boxes.

The **Copy**, **Edit**, **Delete** and **Renumber** menu options act in the same way as their counterparts in the Kinematic Joint Manager.

Flexible Sliders

Simcenter Femap's support for kinematic analysis of mechanical systems is enhanced with the addition of **Flexible Sliders** simulation entities in version 2022.1. **Flexible Sliders** allow analyses to accurately simulate the deformation of curvilinear tracks as assembly components are displaced along them.

This capability includes several parameters for customization such as different slider types, driver loads, and additional friction options. The available slider types allow different constraints for relative rotation of the nodes sliding along the track. This includes spherical, prismatic, cylindrical, and universal types. For driver loads, **Flexible Sliders** can be driven with both external load sets, and forces/displacements applied directly to a specified driver node. Furthermore, for friction, available types include options for validating motion (no friction), infinite friction, and displacement or velocity dependent frictional forces.

When defining **Flexible Sliders** in Femap, multiple options are also available for specifying their required references. Existing group entities can be assigned to the **Flexible Sliders** to define the nodes to be displaced along the track, or the beam elements making up the track. In the absence of group entities, dynamic selection lists can be also be used to assign these entities *on-the-fly*. After creating the **Flexible Sliders**, they can be selectively exported to a SOL 402 analysis with support for specification of single **Flexible Sliders** or a union of **Flexible Sliders**.

In Femap, **Flexible Sliders** are found in the **Simulation Entities** branch of the **Model Info** tree. Or you can access them from the main menu: **Model → Simulation Entities → Flexible Sliders...**

From the **Model Info** tree, right click on **Flexible Sliders,** and select **New** from the menu. This selection opens the NASTRAN Flexible Slider dialog box which is where you define the parameters for the Flexible Slider Joint.

There are four Flexible **Slider Type**s: **0..Spherical**, **1..Cylindrical**, **2..Prismatic**, and **3..Universal**. The **Driver Load** sections contains **Driver Type**s which include **1..Force** and **2..Displacement**. The The **Friction** section has an **Option** drop-down with several friction parameters.

The **Displacement Entities** section allows you select the nodes you want to displace along the flexible slider joint. Select either the from the **Node Group** driop-down or create a dynamic selection list of the nodes (**Node List → Select...**).

The **Slider Trajectory** section is where you define the beam elements that comprise the flexible and curvilinear track. Select either the from the **Beam Group** driop-down or create a dynamic selection list of the beams (**Beams List → Select...**).

When the Flexible Slider is defined, you can export it to an Analysis Set. Select **Model → Analysis.....**. Then in the Analysis Set Manager, expand the tree structure, expand the **Flexible Sliders** branch, and click on the Flexible Slider.
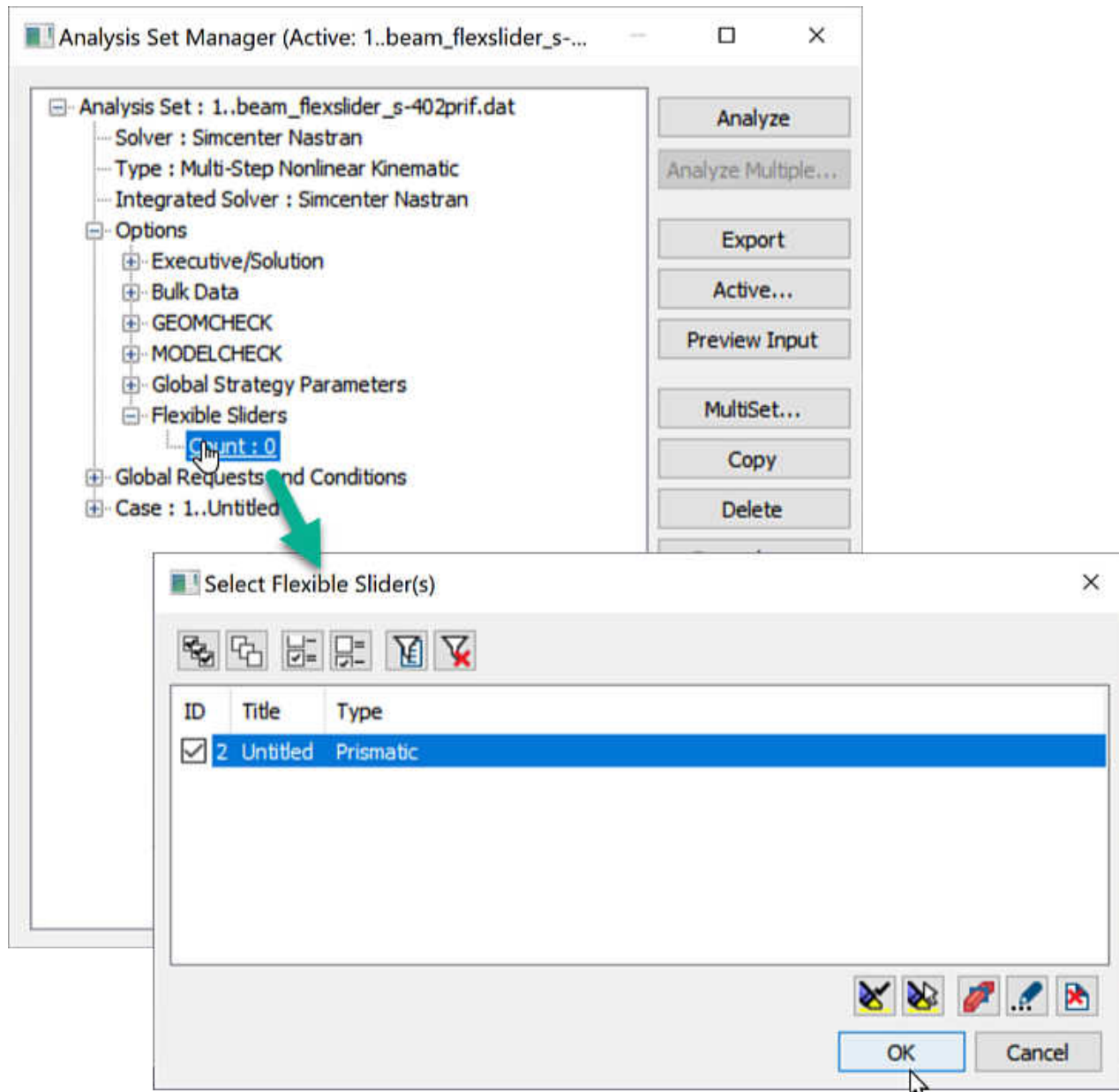
After the analysis is successful, and post processing is completed, the analysis study can be animated showing deformation and stresses in both the flexible path and the car body.

In the animation, the track deforms as the car travels along it.
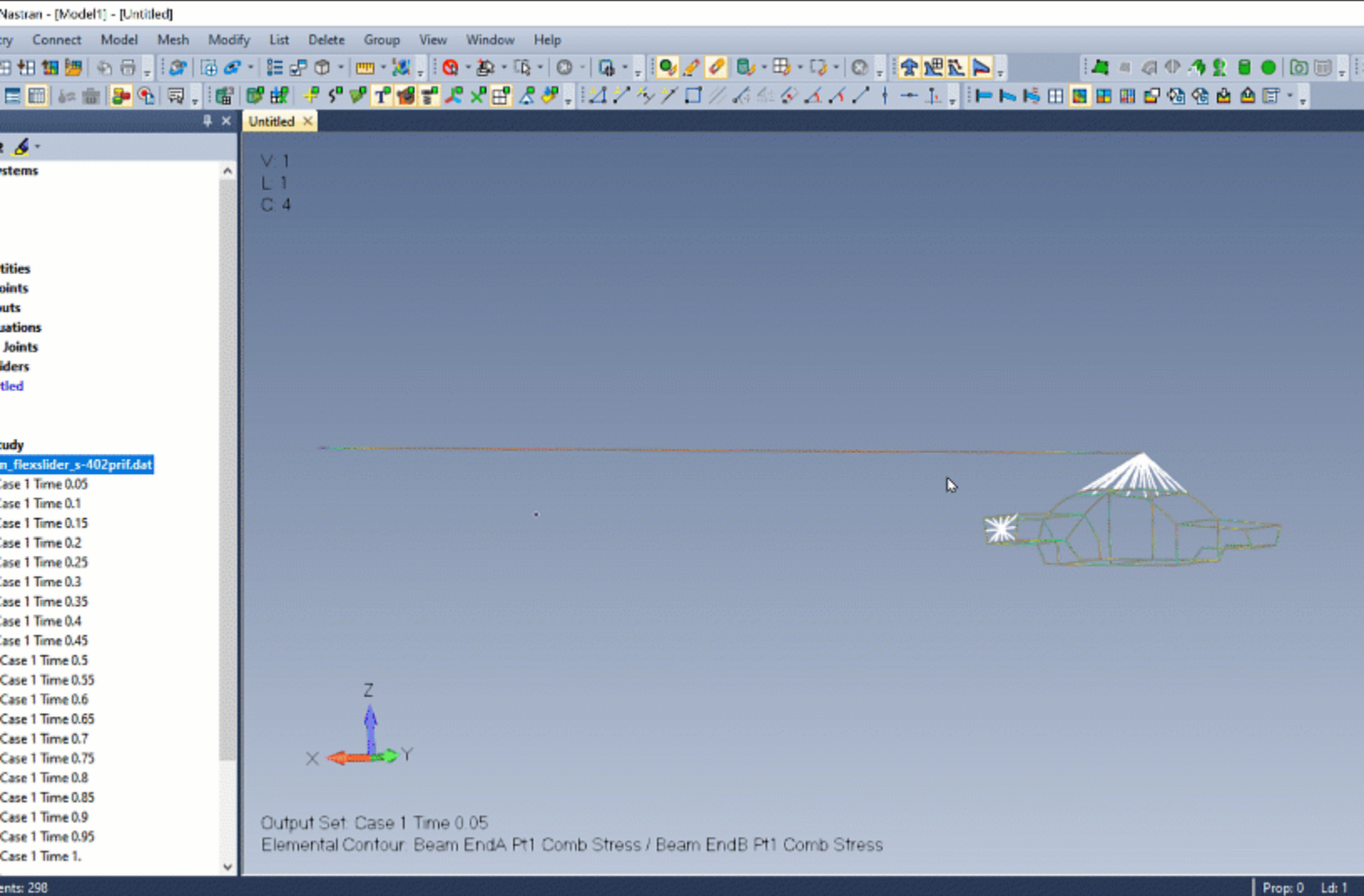
Analysis Program Interfaces

In Simcenter Femap's ongoing mission to provide support for functionality available in Simcenter Nastran, as well as other Nastran solvers, a number of enhancements have been made for version 2022.1. These include:

- **Simcenter Nastran** - Added the ability to request Simcenter Nastran to produce an output file, in particular an *.op2* file, in a format which can then be used by, Femap, and other applications in the Siemens Simcenter Portfolio, including Simcenter 3D, and other commercially available pre and post processors. See **Request Output To .op2 File** for more information.

- **Simcenter Nastran** - Added support for **Tension-Only Quadrilateral Shell Elements**, which have been added to the Multi-Step Nonlinear solution sequence, SOL 401, for Simcenter Nastran. See **Support for Tension-Only Quadrilateral Shell Elements** for more information.

- **Simcenter Nastran** - Added support for a new way to specify control options for the Dynamic Design Analysis Method, often called **DDAM**. See **New Methodology for Control Option Specification in the Dynamic Design Analysis Method (DDAM)** for more information.

- **Simcenter Nastran** - Added new **Solution Monitor** options for monitoring analyses. See **Nastran Solution Monitor** for more information.

For details, see *Section 8, "Analysis Program Interfaces"* in the Femap User Guide.

Request Output To .op2 File

First, it is now possible to instruct Nastran to create an output file, in particular an *.op2* file, which can be used by other applications in the Siemens DI Software Simcenter Portfolio, including Simcenter 3D, from within the Femap User Interface via the Analysis Set Manager. In addition, there is also a preference which can be set to have this be the default behavior.

To have Femap request output be placed into the *.op2* file in this format, use the **Analysis Set Manager** command (**Model → Analysis...**) and select **Output Requests** from the tree structure.
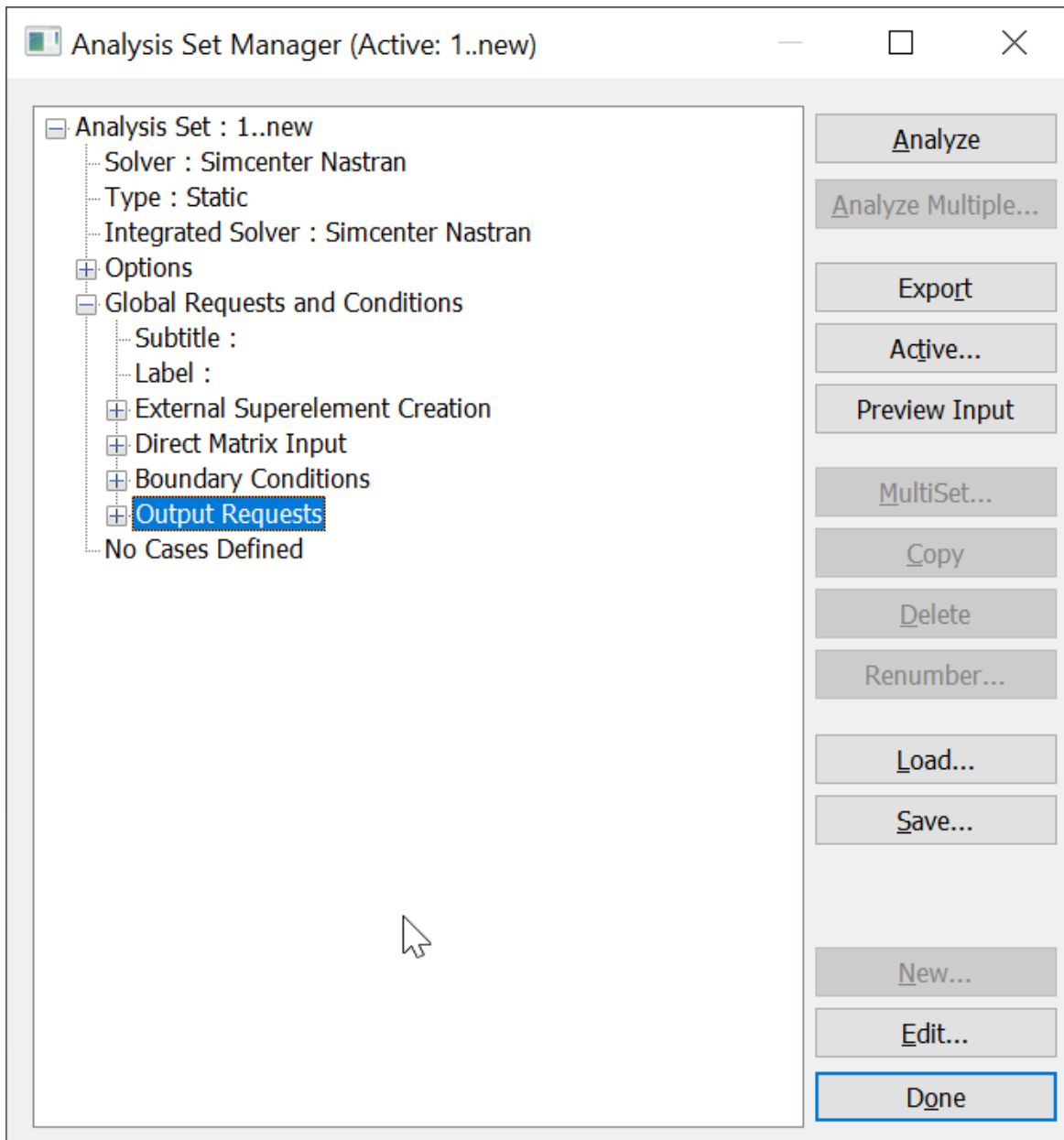
Click **Edit...** to open the Nastran Output Requests dialog box. Select **Simcenter** in the **Results Destination** section. That is all that is needed.

Nastran now knows to produce this type of output by having PARAM,POST set to a value of -2 in the input file.
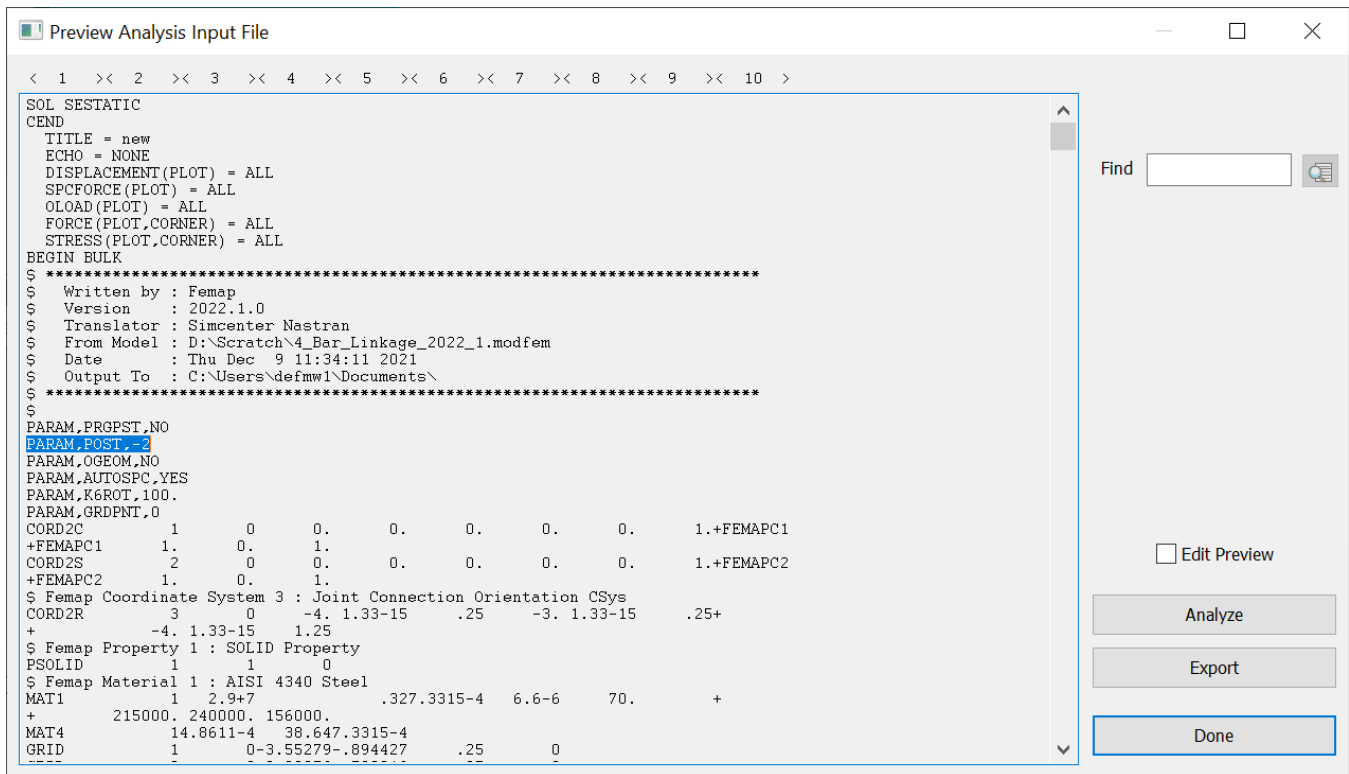
You can also set Femap to always create **.op2** files using PARAM,POST,-2 with the **File → Preferences** command. In the Preferences dialog box, choose the **Interfaces** tab, and set the **Results Destination** option to **Simcenter**.

Some pre and post processors that can read *.op2* files created using PARAM,POST,-1, which is the Femap default, require that the model information also be in the *.op2* file, which is done by setting PARAM,OGEOM to a value of YES. By turning on the **Activate OGEOM Parameter** option on this same tab, all newly created Analysis Sets will have the OGEOM option turned on automatically.

Support for Tension-Only Quadrilateral Shell Elements

Support for Tension-Only Quadrilateral Shell Elements is added in Simcenter Nastran's Multi-Step Nonlinear solution sequence, SOL 401. The Tension-Only Quad will behave like a typical shell element instead of behaving like a shear panel element based on component direction and a compression allowable, both specified by the user, which creates a PSHLPNL entry in the Nastran Input File.

Once the conversion has occurred, the element will continue to act as a shear panel for each Sequentially Dependent subcase following an initial Sequentially Independent subcase. Also, it is now possible to specify a parameter which controls if the tension-only capabilities should be considered for a particular analysis.

In Femap, the behavior of Tension-Only Quad elements is controlled by specifying **Property Extensions** which are available when defining a Plate Property. Select the **Model → Property...** command making sure the **Elem/Property Type...** is set to **Plate**. Choose the **Simcenter/Autodesk Nastran** tab, and then specify which component direction of use for every quad element referencing that property, such as **Use**

X Membrane Stress, **Use Minimum Principal Stress**, etc, along with a **Compression Allowable** value to determine when a shell element will start to behave like a shear panel.
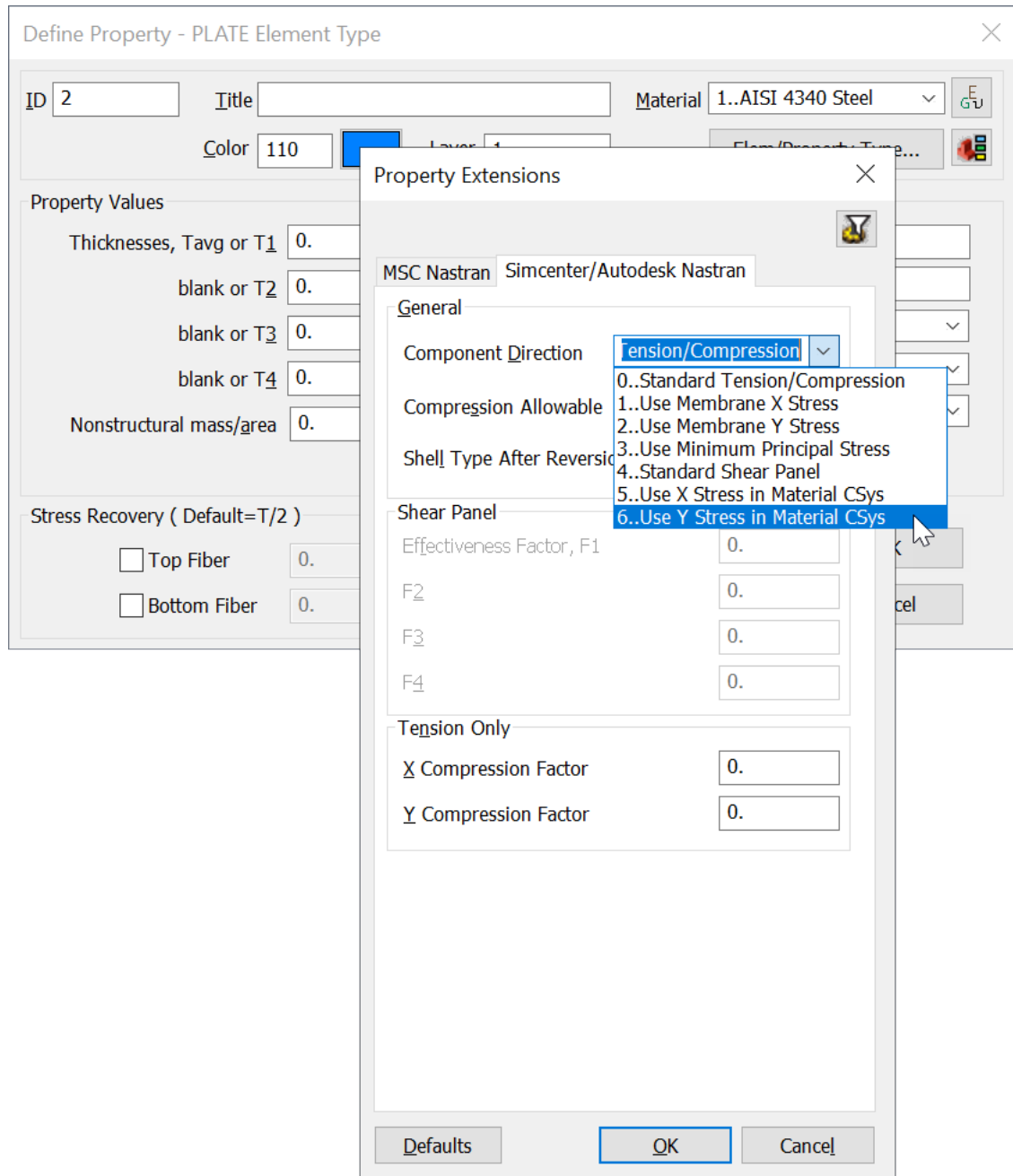
Once converted, the element retains shear panel behavior for the remainder of any sequentially independent subcase and any sequentially dependent subcases which follow.

After the **Property Extensions** are specified, the **TENSOQD** parameter can be specified in the **Analysis Set Manager** to control the behavior of Tension-Only Quads. Select the **Analysis Set Manager** command (**Model → Analysis...**), select **Bulk Data** from the tree structure, and click **Edit...** to open the Nastran Bulk Data Options dialog box.

By default, the ability to have shell elements convert to shear panel elements is active but can be deactivated by turning on **TENSOQD** and setting it to **Tension-only Quad behaves as shell**. In addition, any time Tension-Only Quad Property Extensions are defined, the ESHLPNL entry is automatically added to Case Control of the Simcenter Nastran Input File.

In addition, any time Tension-Only Quad Property Extensions have been defined, the ESHLPNL entry will automatically be added to Case Control of the Simcenter Nastran Input File.
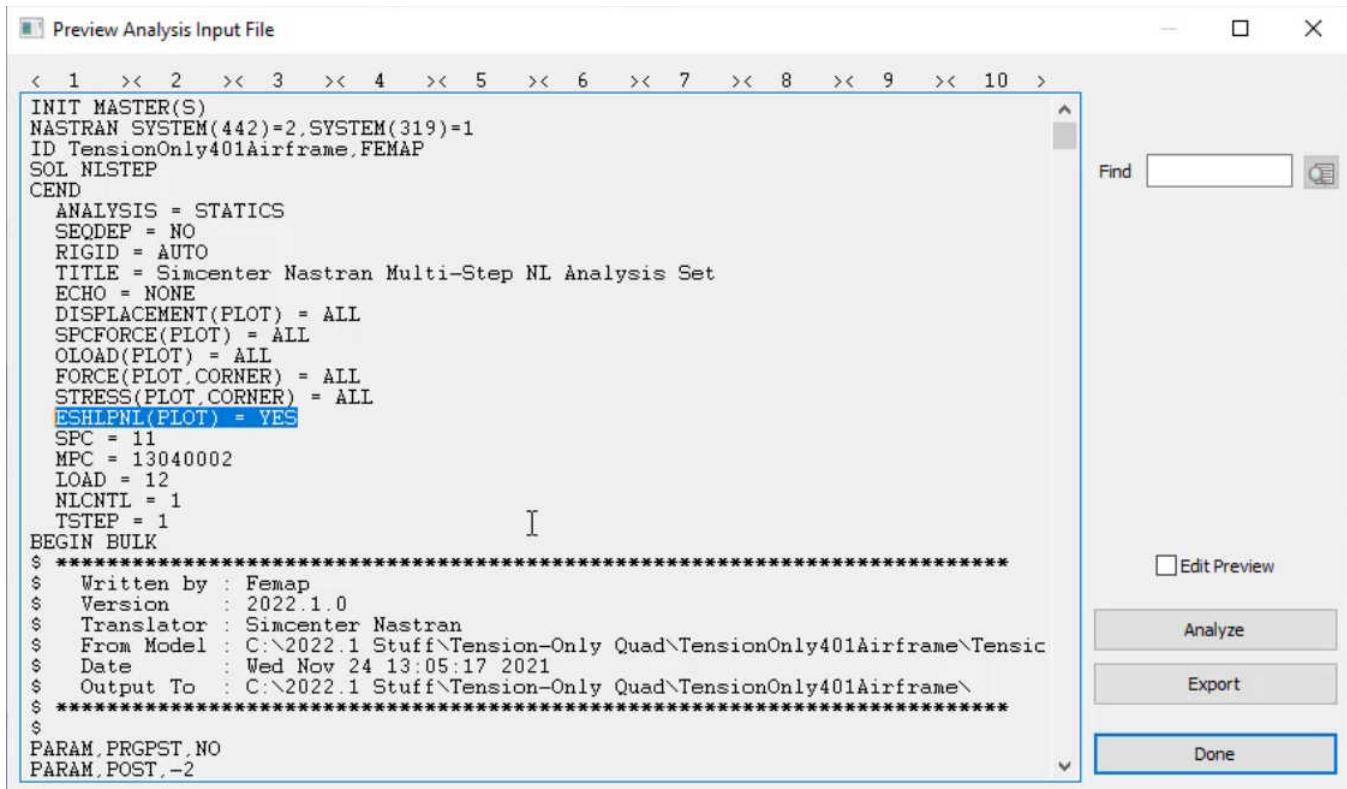
Once the analysis is completed, it is easy to visualize which elements wre converted to Shear Panel Elements by displaying Shear Panel Results. After looking at the shear panel results, it is also possible to look at the shell element results to see which elements were not converted to Shear Panel Elements.

New Methodology for Control Option Specification in the Dynamic Design Analysis Method (DDAM)

support was added for entries in Simcenter Nastran which offer a streamlined approach to specify control options for the Dynamic Design Analysis Method (DDAM). DDAM is a three step process which runs a modal analysis, sends needed results and other inputs specified by the user to the NAVSHOCK Fortran application, then creates Summed Modal Results during the final step.

The Dynamic Design Analysis Method, often called DDAM, is added to Simcenter Femap. DDAM is performed in 3 steps.

1. Simcenter Nastran performs a modal analysis on the structure to calculate the modal participation factors and modal effective mass for each mode.

2. The Naval Shock Analysis FORTRAN program, NAVSHOCK, uses the modal effective masses and natural frequencies calculated by Nastran along with user inputs to compute shock design accelerations.

3. Simcenter Nastran uses the shock design accelerations along with the modal participation factors and natural frequencies to compute the shock response of the structure.

The first thing needed within in Femap to run a DDAM Analysis is an Analysis Set specified to run a Normal Modes/Eigenvalue solution. Select the **Model → Analysis...** command, and click **New...** in the Analysis Set Manager dialog box. In the Analysis Set dialog box, select **2..Normal Modes/Eigenvalue** from the **Analysis Type** drop-down.
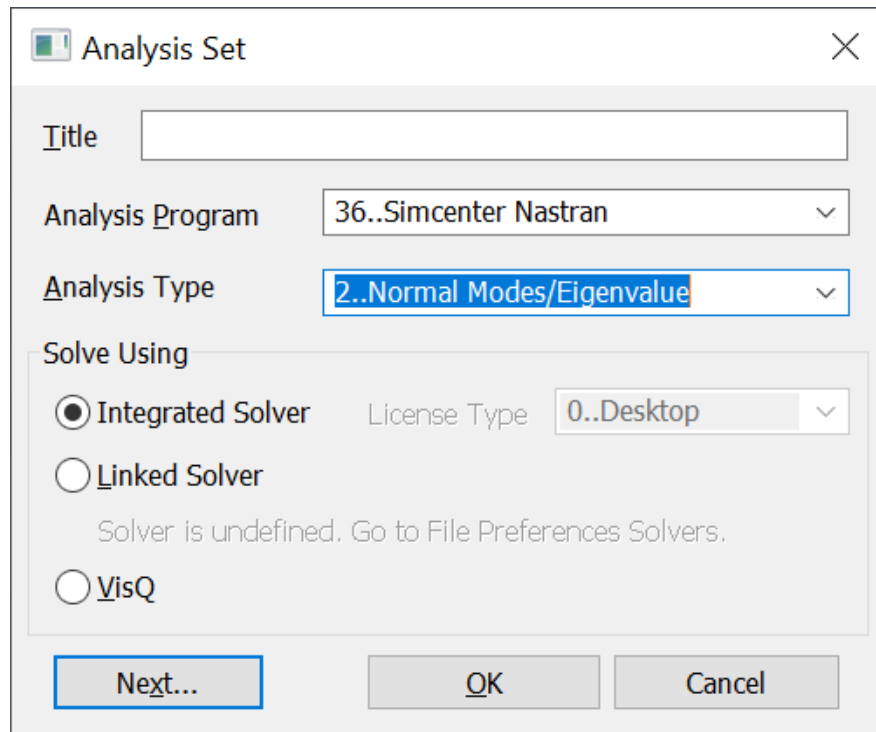
Once that is specified, almost everything else is controlled via the NASTRAN DDAM Solution Options dialog box. Click OK to create the new Analysis Set. Then in the Analysis Set Manager, choose DDAM Options from the tree structure, and click **Edit...**

The first thing you want to do is to check the **Enable DDAM Analysis** option. Currently, if you choose the **DDAM** option in the **Spectrum/Coefficient Options** section, Femap runs DDAM using the legacy method. This method entails creation of multiple external files. All the options in the NASTRAN DDAM Solution Options dialog box are written to a control file which is then sent to the NAVSHOCK program.

To use the new methodology, choose the **DDAMCTR/SPDIR** option, and instead of all the options being written to an external file, they are written to two Nastran entries which are included in the Nastran Input File. Back in the Analysis Set Manager dialog box, click **Preview Input** to see the values are there for those Nastran entries.
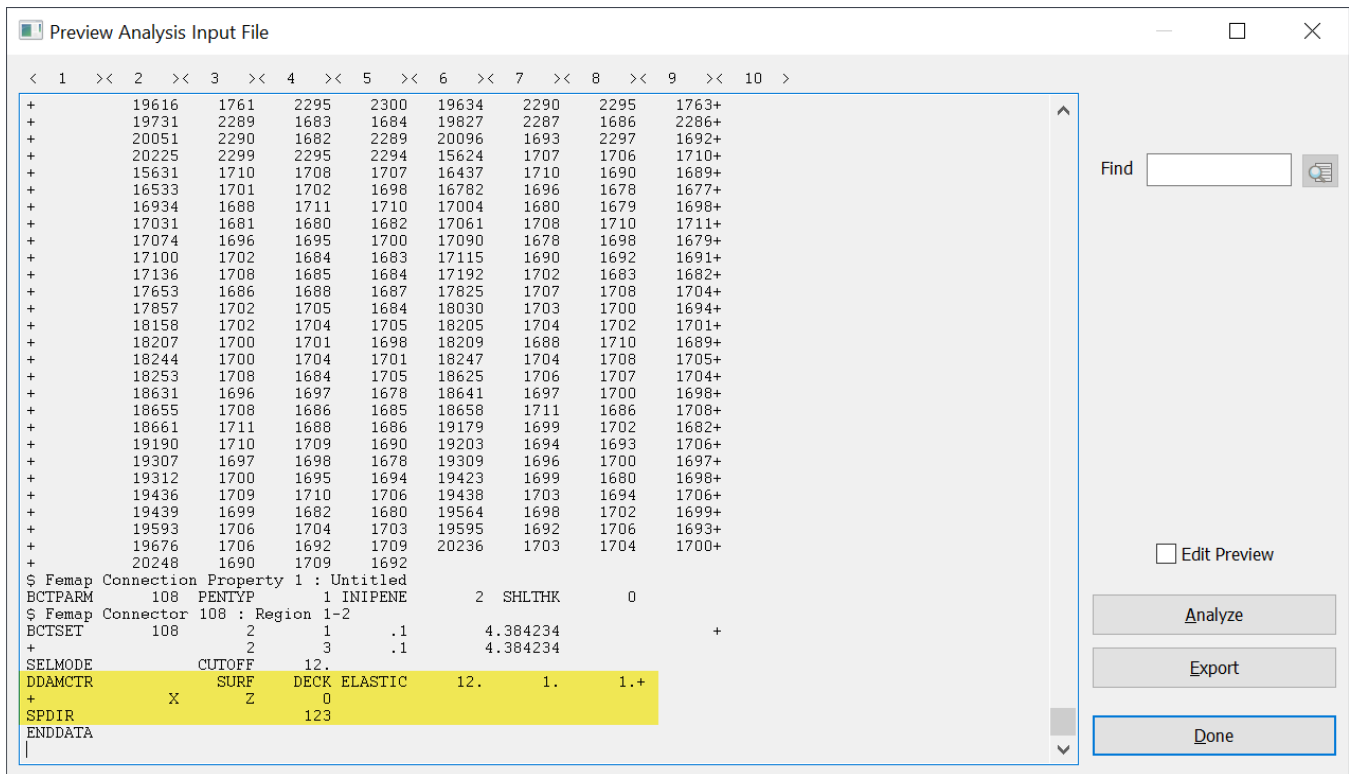
Once the DDAM Analysis is complete, three output sets in the **Model Info** can be explored. They are the **Summed Modal Response** in the X, Y, and Z directions, respectively.
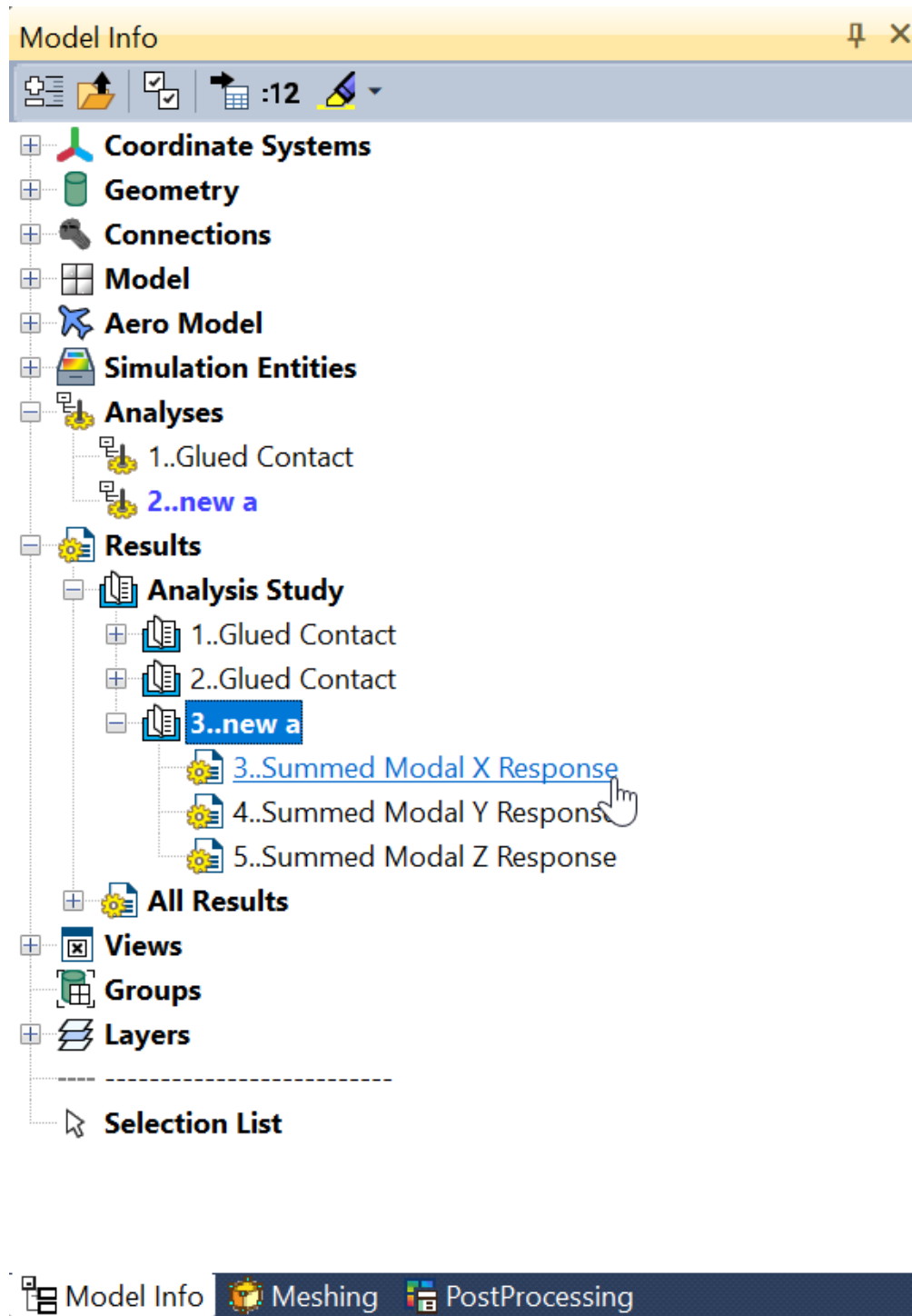
Nastran Solution Monitor

The Simcenter Nastran **Solution Monitor** can now be accessed by Simcenter Femap 2022.1, and allows you to view the results of an analysis and/or review curated output from Simcenter Nastran including information, warning, and error messages recovered directly from Nastran while the analysis is running or after it has been completed.

The **Solution Monitor** is most helpful when used to monitor Simcenter Nastran's Multistep Structural or Multistep Nonlinear Kinematic Solutions Sequences, SOL 401 and SOL 402, thus is the default for those types of analysis. It can also be invoked for other types of analysis by selecting the appropriate option for **Solution Monitor** when specifying **Executive and Solution Options** in the **Analysis Set Manager**.

The reason the solution monitor is so valuable when using SOL 401 or SOL 402 is it provides information such as the time increment and iterations for each step, convergence criteria, cumulative number of iterations, energy evolution, contact/material status, number of nodes in contact, damaged elements, creep elements, and elements which have experienced plastic deformation. Knowing this type of information during a multistep solve can be used to determine if the solution is properly converging and should be allowed to continue or is diverging and various options and/or model setup choices should be reevaluated.

You can access the new **Solution Monitor**. First open the Analysis Manager dialog box (**Model → Anslysis...**), choose **Executive/Solution** from the tree structure, and click **Edit....**

Then in the NASTRAN Executive and Solution Options dialog box, choose **2..Launch SimcenterNastran's Solution Monitor** from the new **Solution Monitor** drop-down list.

Click OK, and then click **Analyze** in the Analysis Set Manager dialog box. The **Simcenter Nastran Analysis Monitor** panel appears docked next to the **Model Info** panel.
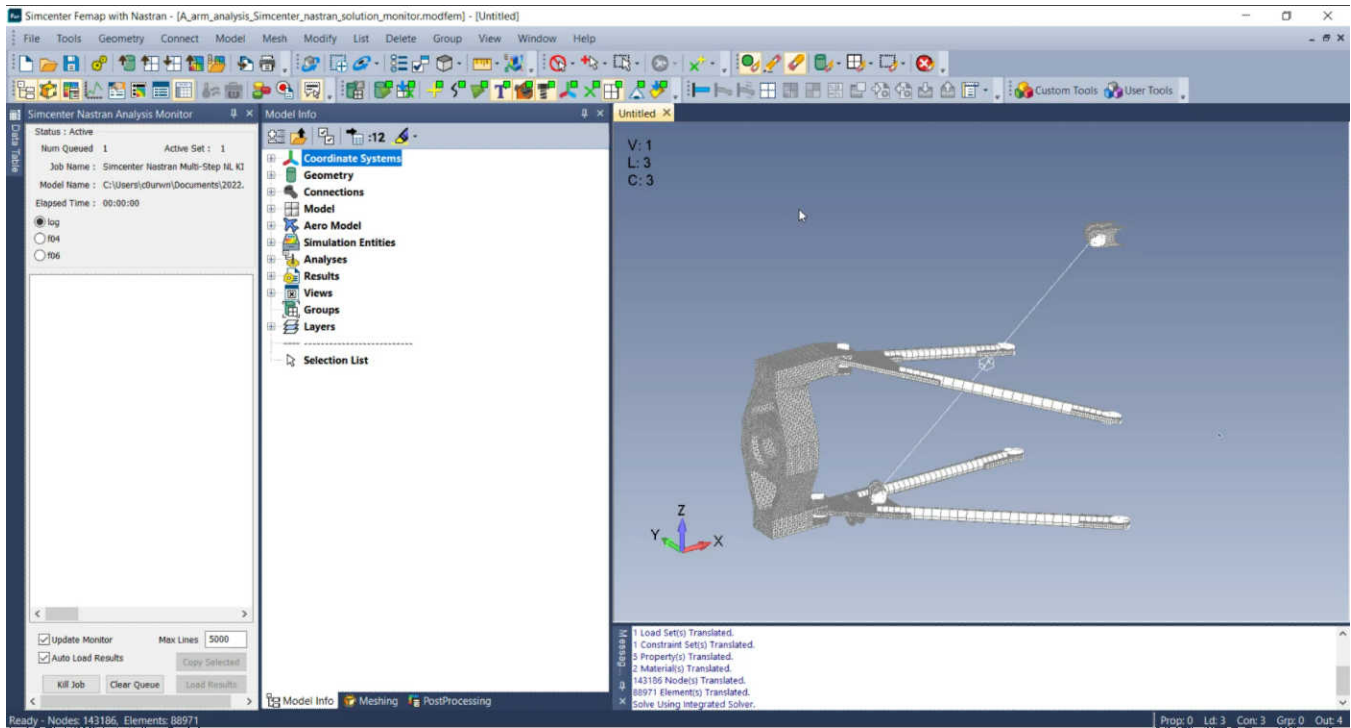
When the analysis starts, the **Solution Monitor** appears indicating the solver has started.
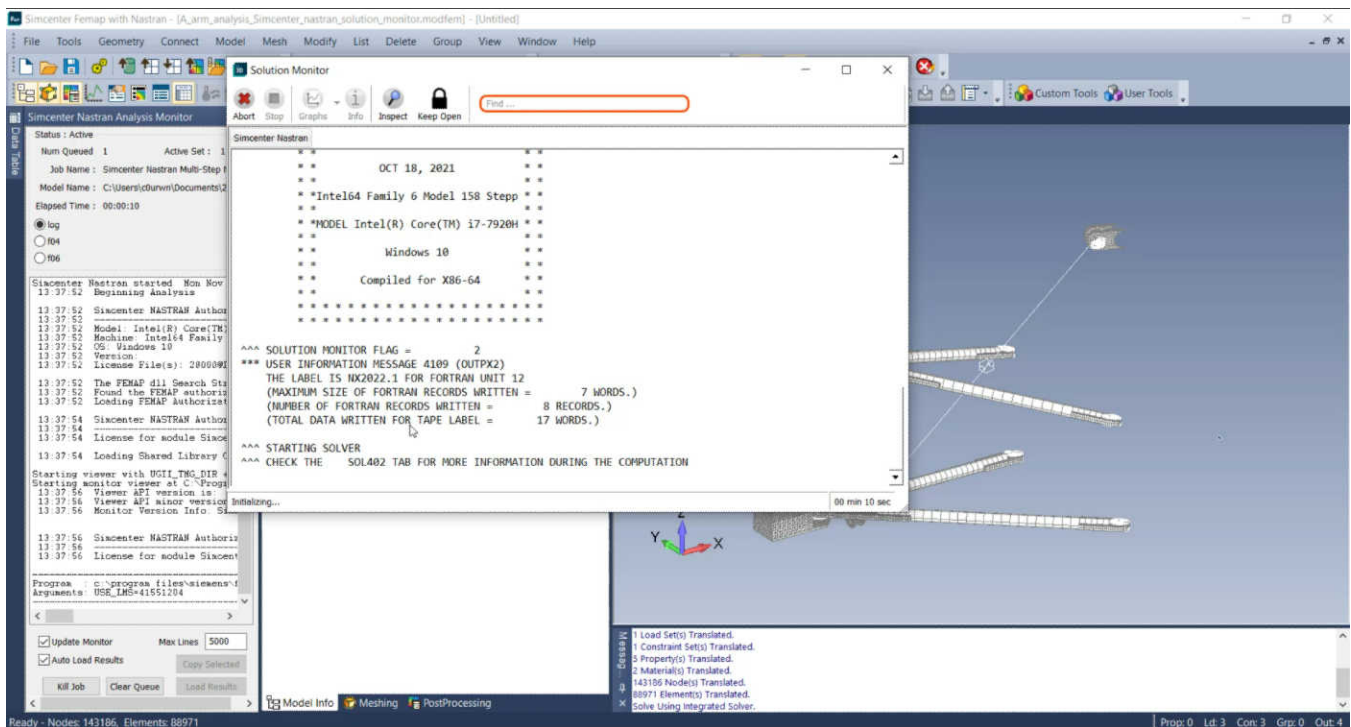
In this example, the analysis is a 402 solution, so there is a **SOL402** tab included. This shows the different time steps requested, and the analysis and iterations it takes for each step. As the monitor goes through and dgets ready to prepare the time steps, you will see a message ---- Sending mesh to

the `Solution Control Monitor ----`. Click the **SC Monitor** icon on the toolbar to open the **Solution Control Monitor** to preview the model mesh.
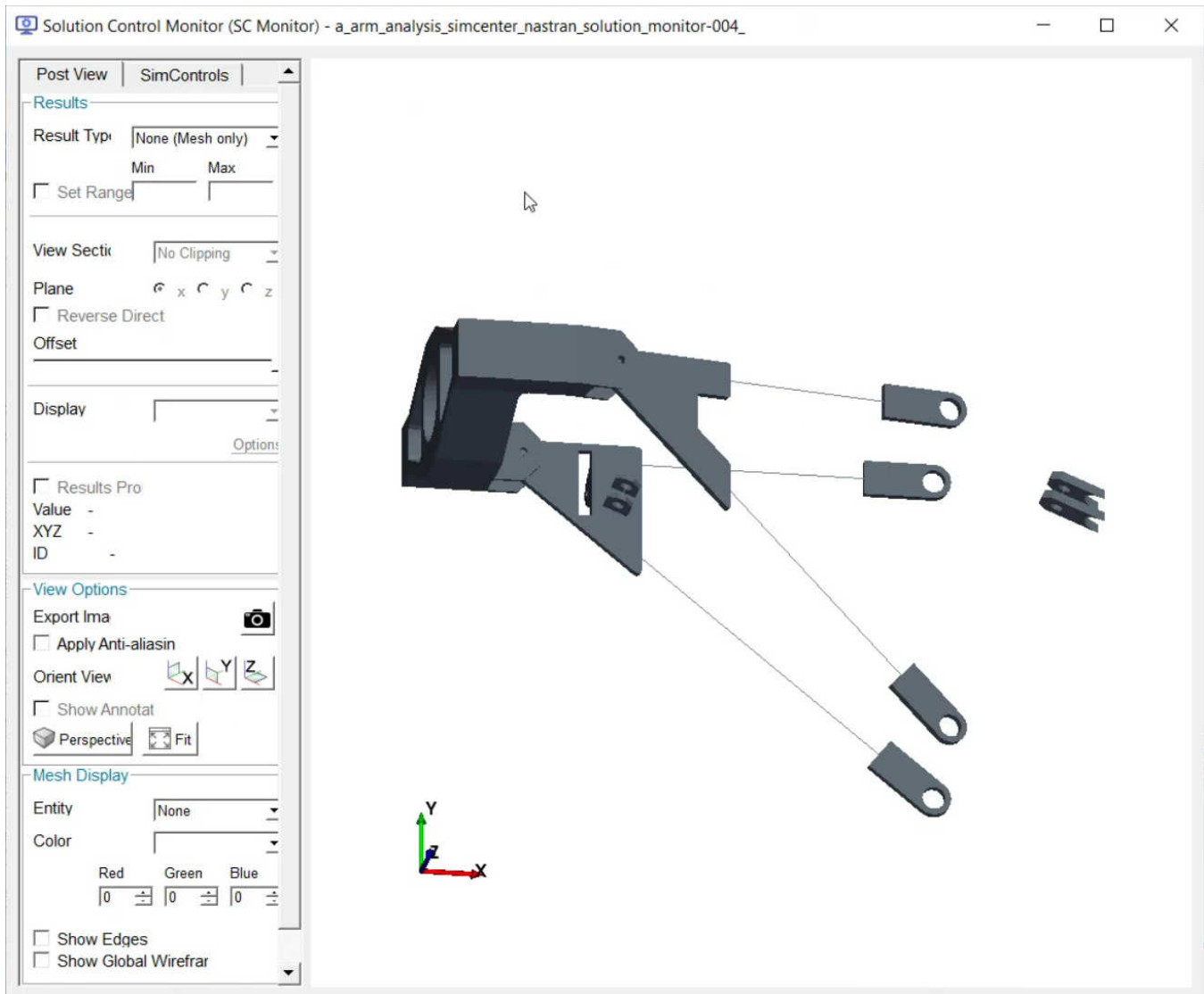
When the analysis is complete, you can review **Results** on the **Post View** tab of the **Solution Control Monitor**. The **Result Type** drop-down includes result types such as **Displacement** and **Displacement Magnitude**. The **View Options** section contains view orientation options and an **Export Image** command to capture the preview in an image file.

The time steps are displayed showing information about each iteration. The next time step begins after the message `-- Time step accepted ---`.
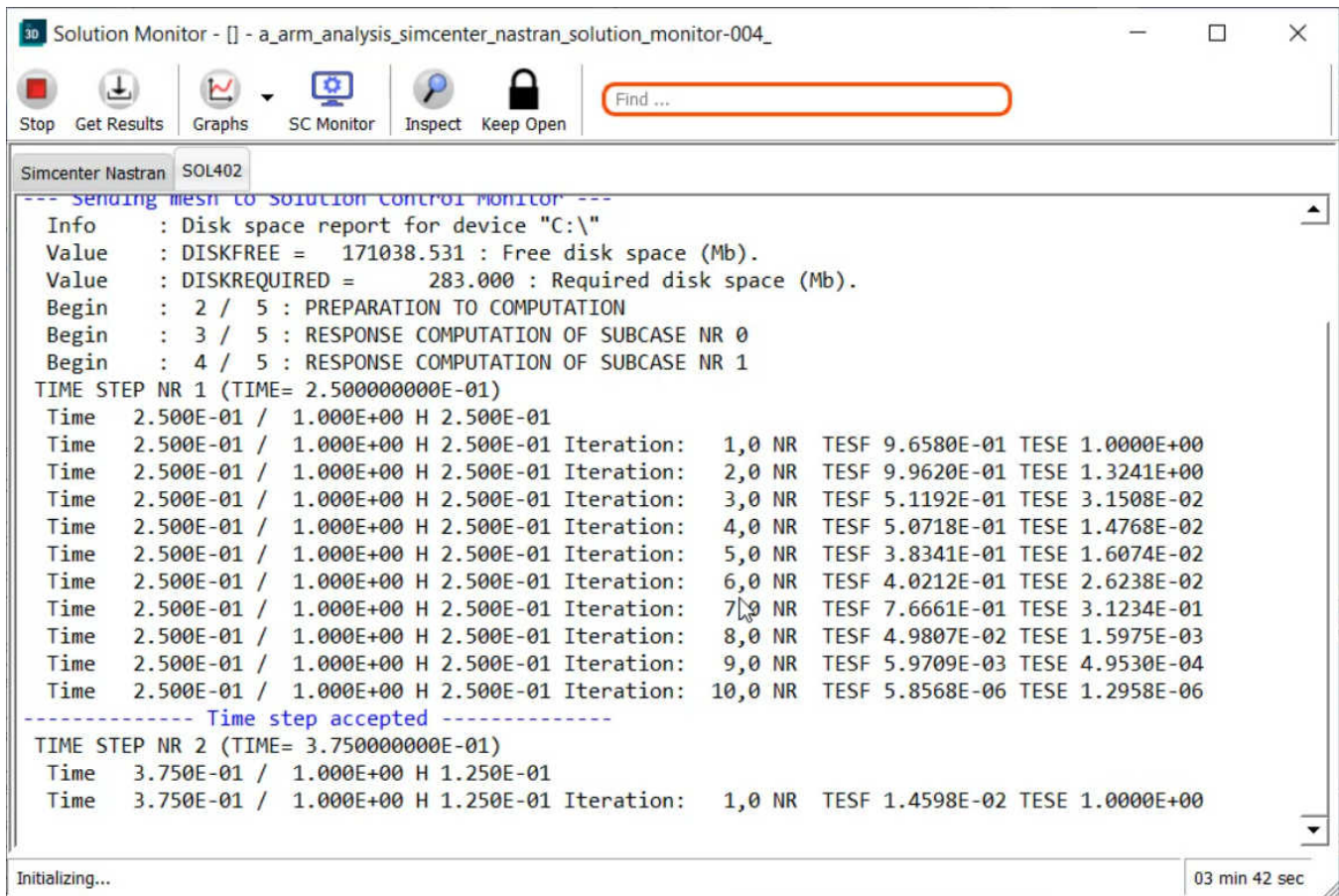
The time steps continue until the analysis is completed. The total number of time steps is displayed at the bottom of the **Solution Monitor**.
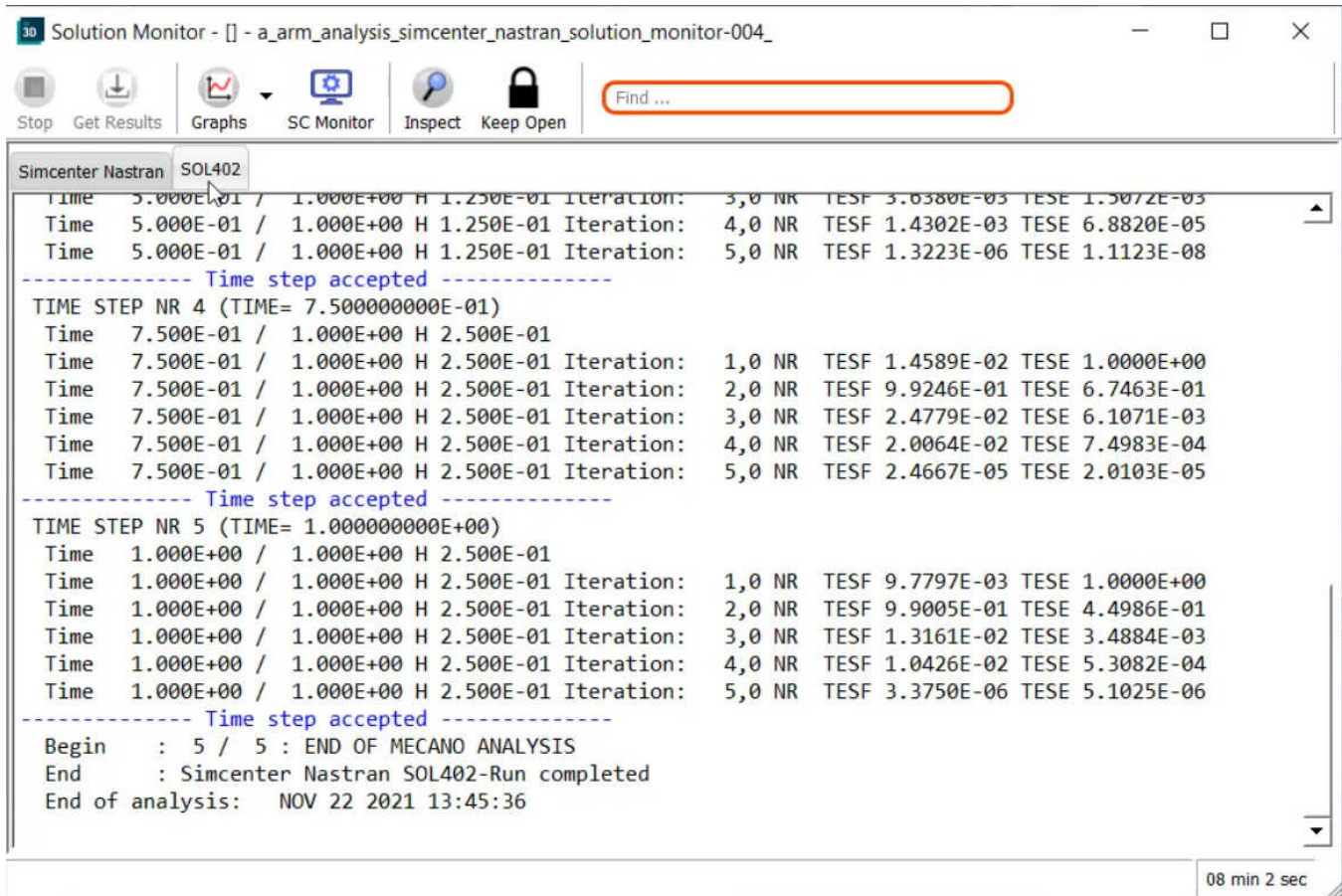
Click the **SC Monitor** icon on the toolbar to open the **Solution Control Monitor** to preview the **Displacement Magnitude**.
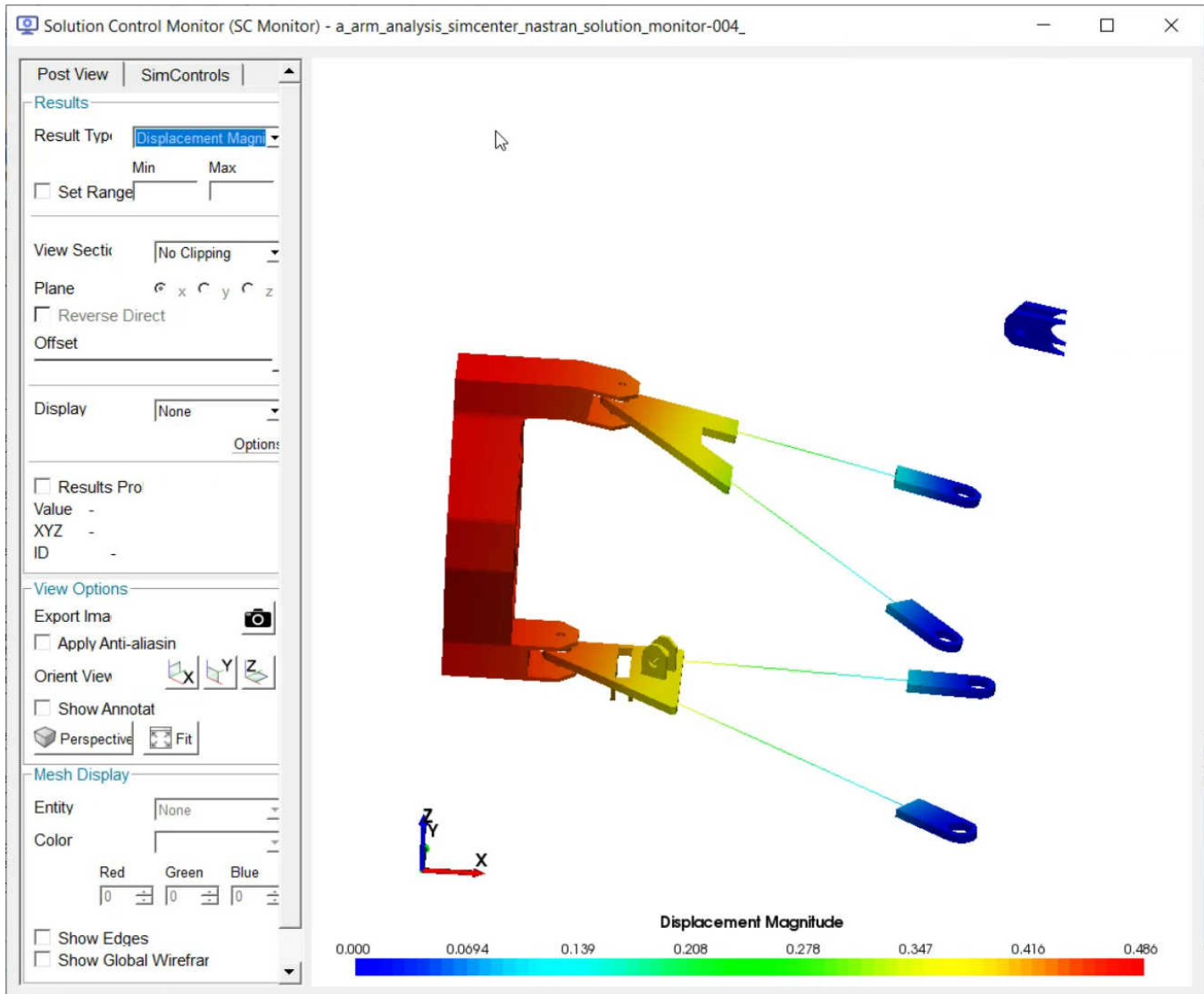
The Solutions Manager also offers several **Graphs** to view **Contact / Material status**, **Convergance** and **Energy**.
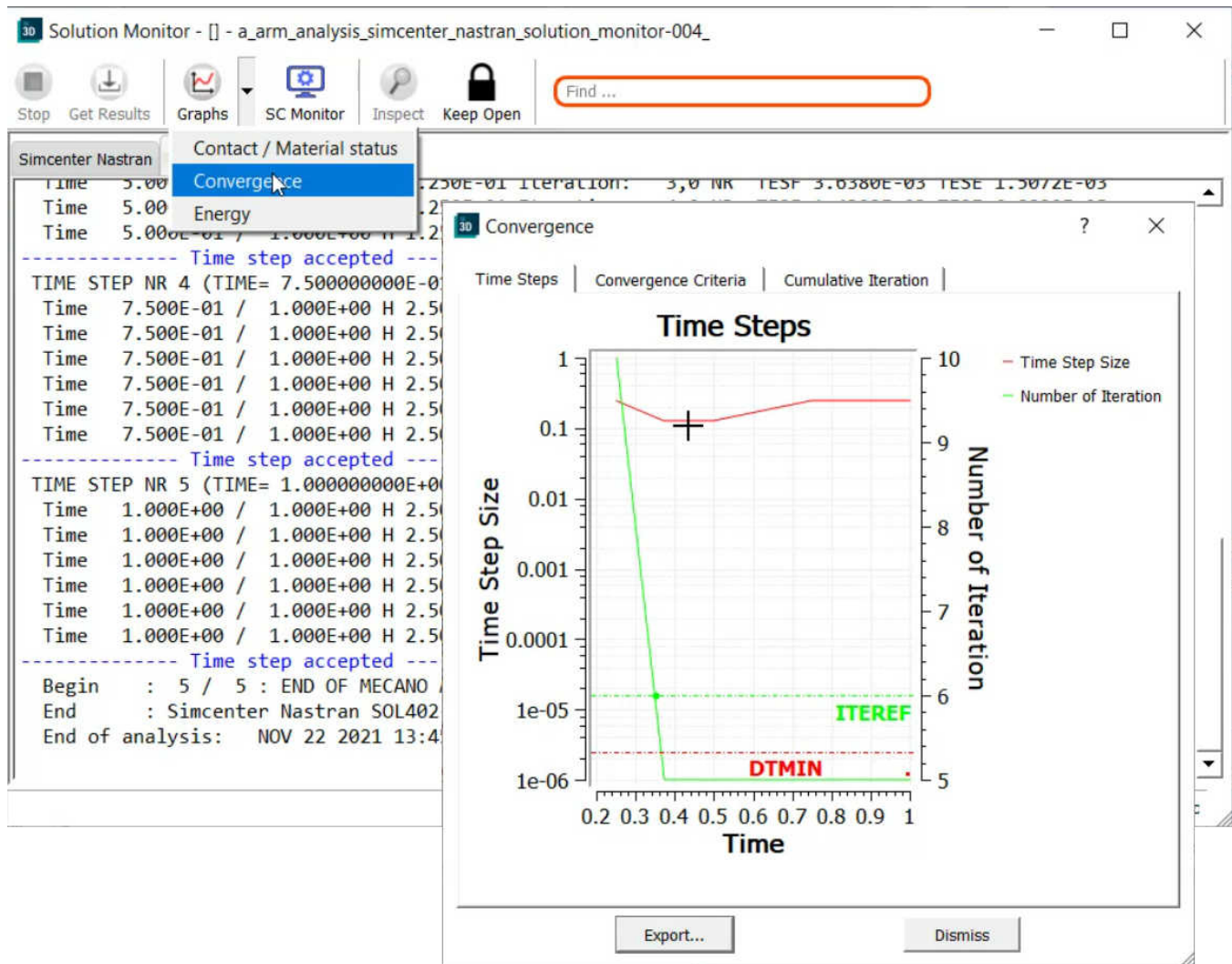
UI/Usability Features

View Entity/Labels Toggle

The standard **Entity Display** Toolbar has been enhanced to include an icon which can be toggled to select if the toolbar is currently controlling the overall display of the various entity types available on the toolbar or the labels for those entity types.

The first Icon now controls Entity or ID.

Evaluate Groups from Model Info

Added the ability to **Evaluate** the rules for any number of selected Groups via the context-sensitive menu for Groups in the **Model Info** pane, which eliminates the need to use the multiple step process when using the **Evaluate** command from the **Group** menu.

If the surfaces are meshed or the mesh is updated, right click the Group in the **Model Info** tree and select **Evaluate** from the menu. Once the view is refreshed, the evaluated group is displayed.
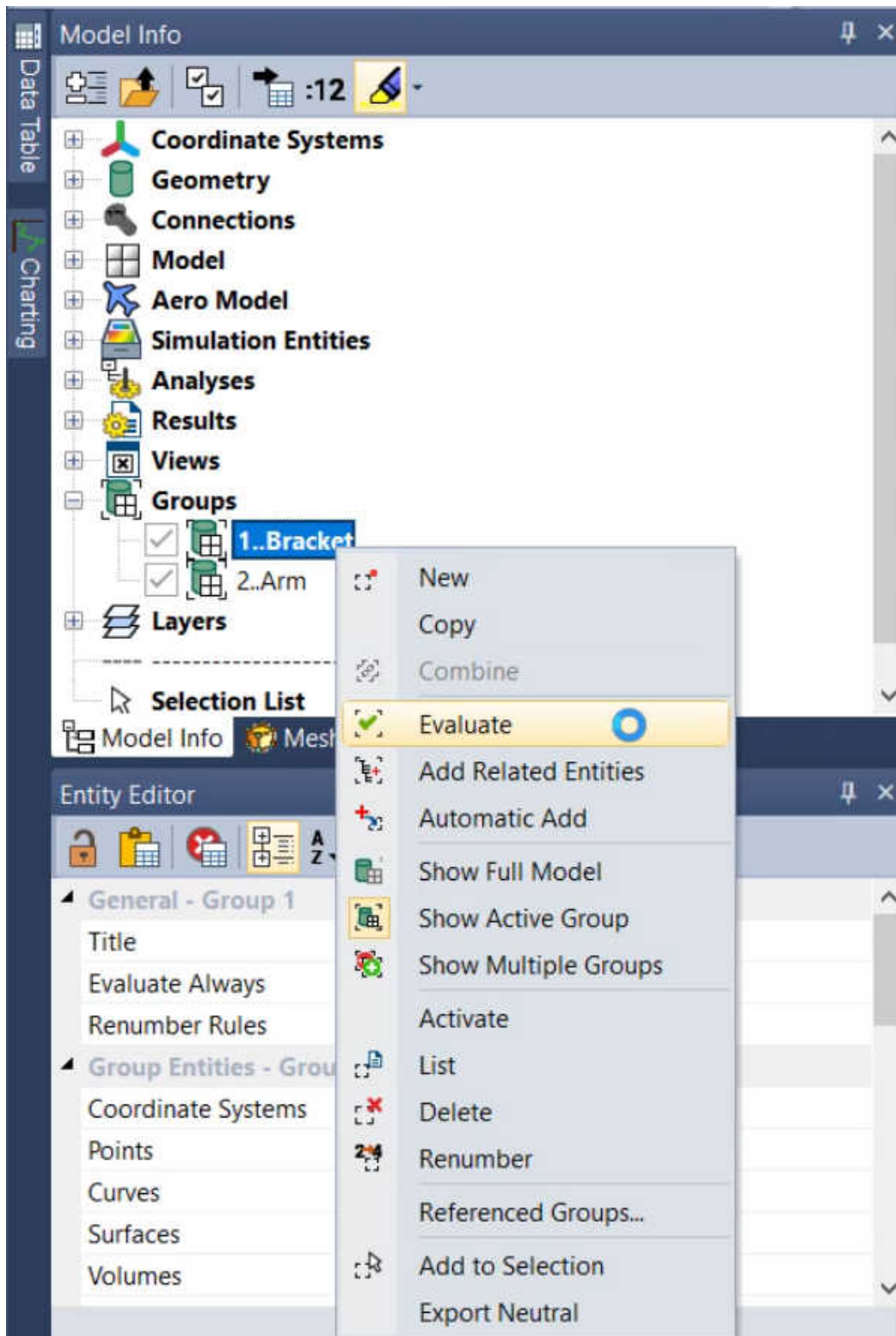
Methods to Select Nodes and Elements on Points, Curves, and Surfaces of Solids

Additional entity selection methods for nodes and elements are now available during standard entity selection. New methods for selection of nodes on the Entity Selection dialog box **Method^** drop-down include **on Points of Solid...**, **on Curves of Solid...** and **on Surfaces of Solid...**.
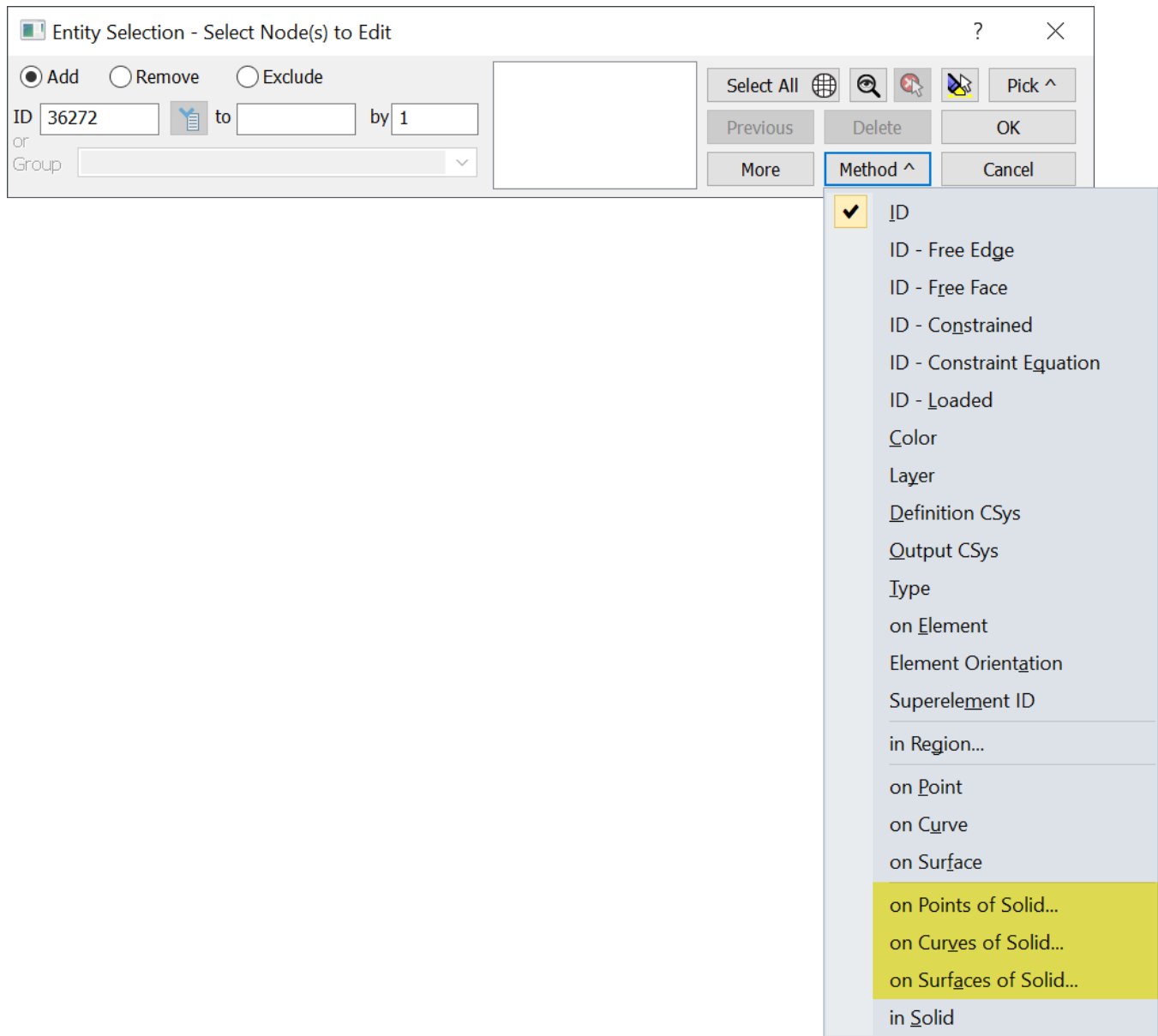
As for elements, new methods on the Entity Selection dialog box **Method^** drop-down include **All Nodes on Solid, on Points of Solid..., on Curves of Solid...** and **on Surfaces of Solid...**.
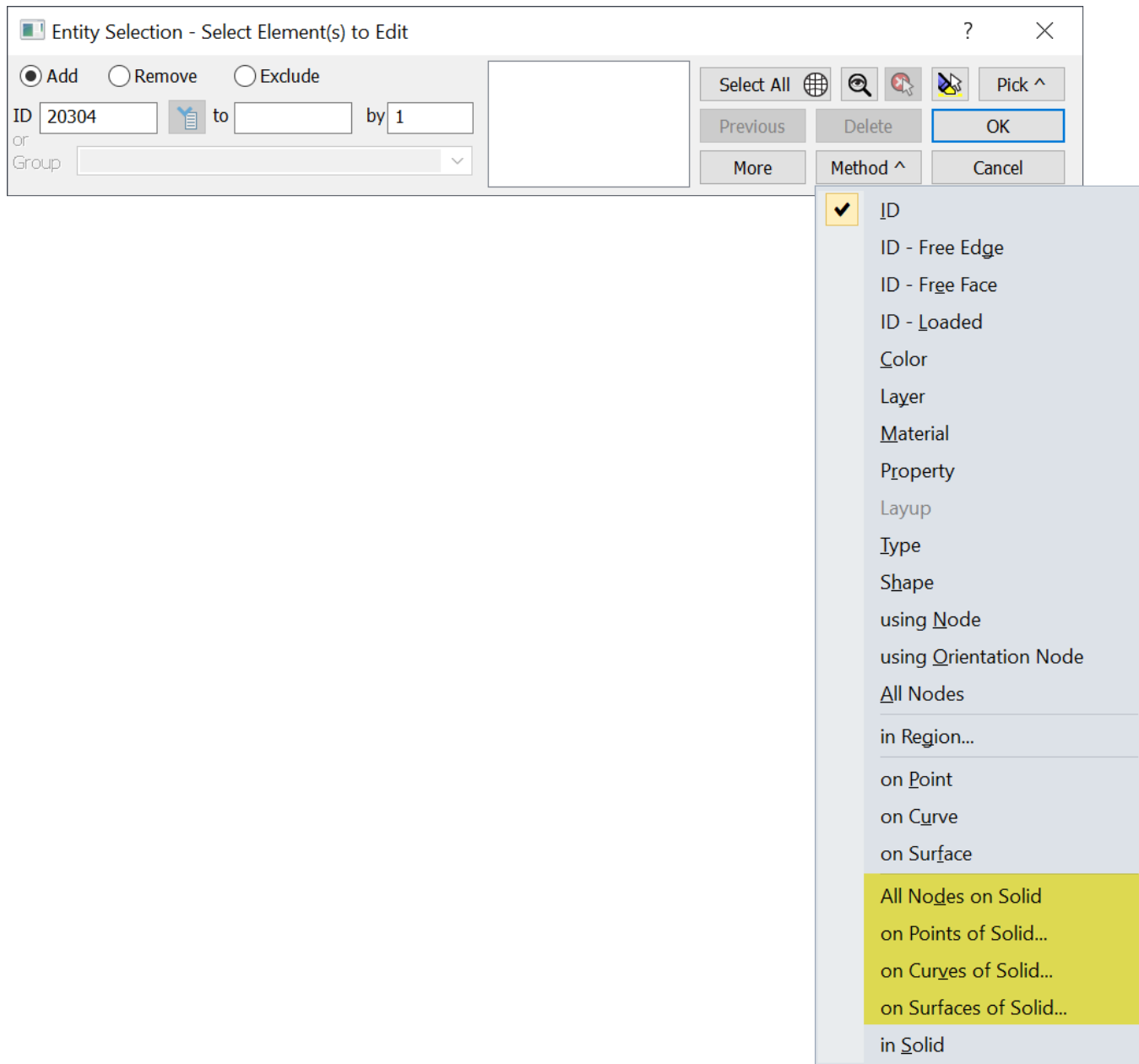
Add Rules to Groups Using the New Node\Element Entity Selection

For Groups, these same selection methods have been made available to use as rules on the **Group** menu for **Nodes** and **Elements**, respectively.
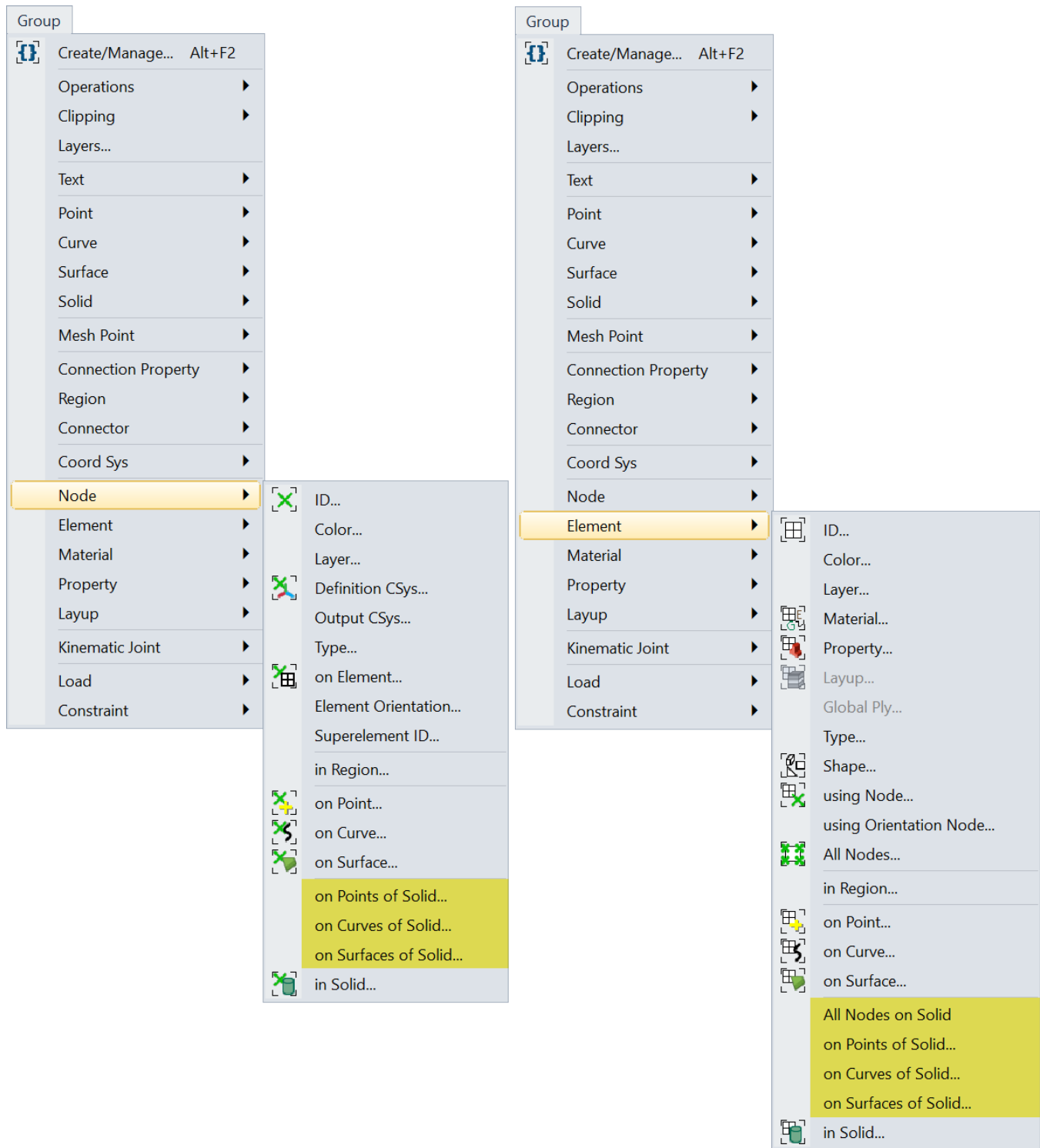
New Help System

The location and format of the Help System for Femap can now be specified via a **Preferences** with the ability to choose the compiled help from previous versions or presented as HTML pages, either accessed

online or locally using the Siemens Help Server application. See **New Help System** for more
information.

New Help System

A big change to Femap's user experience is a new Help System providing the ability to select Online Help
or Local Help.

A new **Help...** button is added to the **File → Preferences** command on the **User Interface** tab of the
Preferences dialog box.

Preferences   ✕

| Geometry/Model | Interfaces | Results | Library/Startup | Color | Spaceball |
| Messages | Views | Graphics | User Interface | Database | Solvers |

Global Options

Language   0..Default   ▾

UI Scaling   0..On   ▾

Menus and Dialog Boxes

☐ Alternate Color Scheme

☑ Autorepeat Create Commands

☐ Remember Dialog Positions

☐ Alternate Accelerator Keys for Views

☑ Ask for Confirmation Before Delete

☐ Show Angles as 0 -> 360

Fast Output Delete   0..Confirm   ▾

Recently Used Files   4

Graphical Selection

☑ Track Mouse Picking   ☐ Pick All Inside

Pick Method   0..Pick Normal   ▾

Snap to   0..Screen   ▾

Tooltip Delay   10   Duration   100

Mouse Interface

☑ Dynamic Zoom Around Cursor Location

☑ Dynamic Rotate Around Cursor Location

☑ Reverse Mouse Wheel Direction

☐ Middle Button Click for OK

☐ Shift for Pan, Ctrl for Zoom

   Wheel Factor for Dynamic Planes   1.

Help...

Meshing Toolbox

☑ Expand Active Tool Only

AutoRemesh   0..On   ▾

Dockable Panes

☐ Animate Fly-Out

☐ Captions Always on Top

☑ Alternate Docking Symbols

Model Info

Max Entities   2000

☑ Create Automatic Titles

Show Entities Defaults

◯ Highlight

◉ Transparent Highlight

◯ Show Selected Only

☐ Show Labels   ☐ Normals

☐ Autoscale

Highlight Color   24600   [▭]

International Localization / Clipboard

☑ Use Region Decimal/List Symbols

☑ Ignore Delimiters if Pasting Tabs

Toolbars

Save Layout...   Load Layout...

Reset User Interface...

Reset Dialog Positions

Reset All   OK   Cancel

Clicking it opens the Help Options dialog box. This dialog controls which source is used to display help content. You do not have to exit Femap for the change to take effect.
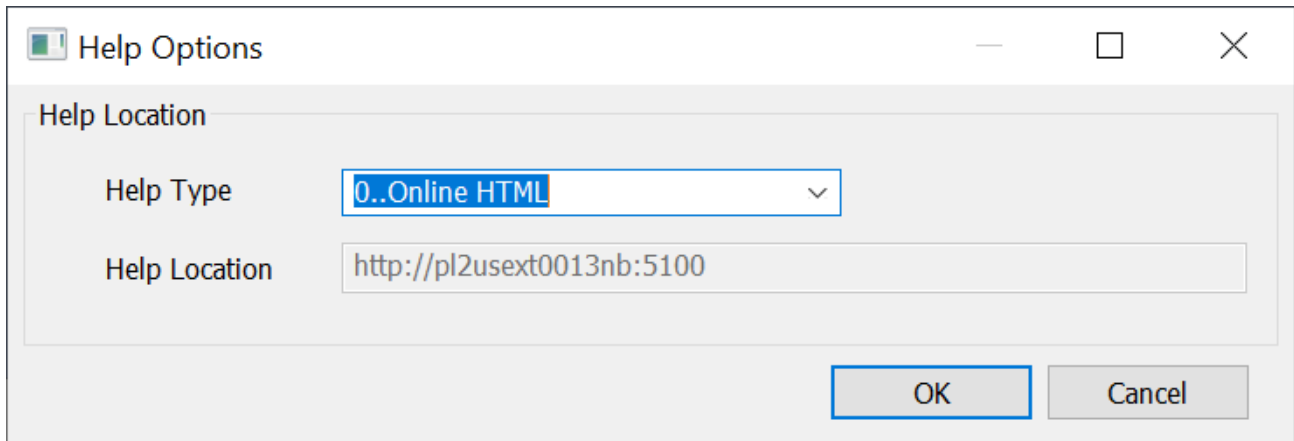
The options in the **Help Location** section are used to select the help system used by Femap.

Help Type

Here, you select which help system to use.

0..Online HTML

Activates online web help. Help is hosted on the Siemens Support Center website - This is the default source when you access Femap Help from the product. This is the web-served HTML help that you view in an internet web browser.

> Note:
>
> Online help for all Siemens software is available on Support Center: **https://www.plm.automation.siemens.com/locale/docs/**

1..Local HTML

Activates locally installed web help. This help system requires you install the Siemens PLM Documentation Server web server application - By installing the Siemens PLM Documentation Server, you can host online help for Femap and other Siemens Digital Industries Software products on your local host or intranet. This installs the HTML help collection on an internal server or a standalone computer. It provides the same content and is also displayed in a web browser. When deployed on your local intranet, the Siemens PLM Documentation Server can host online help for multiple languages and multiple Siemens Digital Industries Software products. It provides a fast, powerful, server-based search engine.

> Note:
>
> To use this option, you must install the Siemens PLM Documentation Server and the Femap help, which are installations separate from the installation of Femap.

> If you do not have user privileges to access the public internet, and if your company provides access to locally installed help, select this option and enter the URL of the local server in the **Help Location** field.

Help Location

Activated when **1..Local HTML** is selected as the **Help Type**. Enter the URL of the local server in this field to use locally istalled help.

API

Overall Functionality

- Updated API methods to be able to pass Variants or single valued variables dimensioned as BOOLEAN, INT2/INT4/INT8, or REAL4/REAL8 to input array arguments if all values of the array are supposed to contain the same value. Previously numeric constants and dimensioned array variables were supported, but not the new combinations.

New and updated API Objects and Attributes

- Added Kinematic Joint and Joint Connection (feJoint) Object to the API.

- Added color, layer, title, type, JointBehavior, G1, G2, PID, HasCID1, CID1, HasCID2, CID2, HasG3, G3, KT, KR, CF, FR0, LR, LX, LY, LZ, RegularizationOpt, TOL, KCF, LIBL, PITCH, OPT, HasSPR1TRA, K1T, K1TTID, SK1TTID, HasSPR1ROT, K1R, K1RTID, SK1RTID, HasDAM1TRA, B1T, B1TTID, SB1TTID, HasDAM1ROT, B1R, B1RTID, SB1RTID, JointLoc1, vJointLoc1, JointLoc2, vJointLoc2, JointLoc1_x, JointLoc1_y, JointLoc1_z, JointLoc2_x, JointLoc2_y, JointLoc2_z, HasTarget1List, HasTarget2List, Target1SetID, Target2SetID, Target1Type, Target2Type, CurveExpansionOpt, SurfExpansionOpt, and ElemExpansionOpt to the Kinematic Joint and Joint Connection Object

- Added Flexible Slider (feFlexibleSlider) Object to the API.

- Added color, layer, title, NasTYPE, NasNGRPID, NasBGRPID, NasSGID, NasPROJ, NasHasNLIM, NasNLIM, NasDTYPE, NasDGID, NasDVAL, NasDTID, NasFOP, NasCF, NasTOL, NasKCF, HasNodeList, HasElemList, NodeSetID, and ElemSetID to Flexible Slider Object

- Added AddToCopiedEntityGroups and CopyMeshSize to the CopyTool Object

- Added NasBulkTENSOQD, NasBulkTENSOQDval, NasSolutionMonitorOpt, NasMsNLKGlobalRESXTRP, NasMsNLKGlobalBOLTMETH, NasMsNLKGlobalPLSHSOL, NasDdamSpdirCSID, NasDdamSpdirCOMP, NasDdamFileUnit, NasMsnlkJCON, NasMsnlCntPLSHUT, and NasMsnlkCnt2PLSHUT to Analysis Manager Object

- Added VectorPressure, vPressureDirection, and PressureDirection to the LoadMesh Object

- Added IsSequence to the LoadSet Object.

New and Updated API Methods

- Added GetReferenceSet, ClearReferenceSet, PutReferenceSet, GetMotionSet, ClearMotionSet, and PutMotionSet methods to the Kinematic Joint and Joint Connection Object

- Added GetComponentSet, ClearComponentSet, and PutComponentSet Methods to Flexible Slider Object

- Added PutFLXSLISetID, GetFLXSLISet, ClearFLXSLISet, and PutFLXSLISet to the Analysis Manager Object

- Added Solid to MeshPoint Object

New and Updated Global Variables

- Added Pref_OnlineHelp_Mode to set Help Type preference in the Help Options dialog box accessed via User Interface tab of File, Preferences.

- Added Pref_OnlineHelp_Path to set directory path for Help Location preference in the Help Options dialog box accessed via User Interface tab of File, Preferences.

- Added Pref_SC3DResultsDest to specify Results Dest preference on the Interfaces tab of File, Preferences.

- Added Pref_nastran_param_ogeom to specify Activate OGEOM Parameter preference on the Interfaces tab of File, Preferences.

The following functions have been added or updated:

- feCheckCoincidentNode5

3. What's New - version 2021.2

User Interface

General

- Added an **All Nodes on Solid** option to the **Method** menu of the standard entity selection dialog box when selecting elements. When used, any element which has all its nodes associated to the selected solids will be selected.

- Updated selection of Aero Panel/Body, Aero Property, Aero Splines, and Aero Control Surface entities to use the picking technology activated by the **New Picking** option on the **Graphics** tab of the **File → Preferences** command, which takes advantage of **OpenMP** to improve performance.

Menu

- Added the **Model → Simulation Entities → Design Equation** command. See **Simulation Entities** for more information.

- Added the **Mesh → Bodies** and the **Mesh → Mesh on Mesh** commands. See **Meshing** for more information.

- Added the **Delete → Model → Design Equation** command. See **Simulation Entities** for more information.

- Added the **Group → Element → All Nodes on Solid** command. See **Groups and Layers** for more information.

Model Info tree

- Added **Design Equation** to the **Simulation Entities** branch which allows for management, editing, listing, deleting, and renumbering of monitor points and matrix inputs, respectively. For monitor points, the color and layer may also be changed.

Meshing Toolbox

- **Geometry Editing** tool - Added a **Projection Mode** drop-down when **Operation** is set to **Point to Edge**. Choose from **Normal**, which performs a normal projection of the selected point to the selected edge; **Vector**, which allows specification of a vector to project the selected point along to the selected edge; or **Interactive**, which is has the user select a point, select an edge, then specify a vector to project along.

- **Geometry Editing** tool - Added a **Projection Mode** drop-down when **Operation** is set to **Project Curve**. Choose from **Normal**, which performs a normal projection of the selected curve(s) onto the selected surface(s) or **Vector**, which allows specification of a vector to project the selected curve(s) along to the selected surface(s).

- **Mesh Sizing** tool - Added **Set Size** to the **Operation** section which allows the user to specify a **Mesh Size** to define the number of elements on a curve. In addition, changed **Set To** in the **Operation** section to **Set Number** which allows the user to specify **Number of Elements** on a curve.

Mesh Control Explorer

- Added **Show Edges that are Adjacent to Surface with No Imprinted Edges** and **Show Surfaces that Pierce Other Surfaces with No Imprinted Intersections** tools to the **Mesh Control Explorer**. Also, changed the position of the icons in the pane to group tools used to do similar operations together.

Show Edges that are Adjacent to Surfaces with No Imprinted Edges

This tool is used to determine if edges of any number of surfaces are adjacent to the face of a surface, but there is no imprinted edge where an edge(s) of the surface(s) meet the face of another surface (T-Junctions) then lists them to the list area. When highlighting is enabled, which is the default, all *Surface Face-to-Surface Edge* combinations which are found by the tool will also be highlighted in the graphics window. For a single combination, this is listed as "Surface ID > [Surface Edge (Curve) ID]". For combinations where edges from multiple surface intersect a single surface, this is listed as "Surface ID > [Surface Edge (Curve) ID A, Surface Edge (Curve) ID B, etc]".

To potentially create imprinted edges on the surface using the surface edges, select a line in the list area, then right-mouse click and choose the **Imprint Edges** from the context-sensitive menu. To attempt to create imprinted edges on all of the *Surface Face-to-Surface Edge* combinations currently in the list area, select any line from the list area, then right-mouse click and choose the **Imprint Edges - All** command from the context-sensitive menu. In addition, if you would like to have the imprinted edges always be extended to the extents of the surface in both directions, activate the **Extend Imprints** toggle on the context-sensitive menu.

List simply lists the surface ID and all curve IDs on the selected line to the **Messages** window, while **Show All** highlights found by the tool using the current options specified on the **Show When Selected** icon menu.

Show Surfaces that Pierce Other Surfaces with No Imprinted Intersection

This tool is used to determine if any number of surfaces pierce through another surface but there are no imprinted curves at the location where the surfaces intersect., then lists them to the list area. When highlighting is enabled, which is the default, all *Surface(s) piercing another Surface without Imprinted Intersection(s)* combinations which are found by the tool will also be highlighted in the graphics window. For a single combination, this is listed as "Surface being pierced ID > [Piercing Surface ID]". For combinations where multiple surfaces are piercing a single surface, this is listed as "Surface being pierced ID > [Piercing Surface ID A, Piercing Surface ID B, etc}".

To potentially create imprinted edges at the intersections of surface(s) which pierce each other, select a line in the list area, then right-mouse click and choose the **Imprint Intersections** from the context-sensitive menu. To attempt to create imprinted edges on all of the surfaces which pierce each other currently in the list area, select any line from the list area, then right-mouse click and choose the **Imprint Intersections - All** command from the context-sensitive menu. In addition, if you would like to have the imprinted edges always be extended to the extents of the surface(s) in both directions, enable the **Extend Imprints** toggle on the context-sensitive menu.

List simply lists the curve on the selected line to the **Messages** window, while **Show All** highlights all of the curves found by the tool using the current options specified on the **Show When Selected** icon menu.

Meshing

- Added the **Body/on Mesh** option to the **Mesh → Geometry → Surface** command. When selected, surfaces are meshed with the same mesher used by the new **Mesh → Bodies** and **Mesh → Mesh on Mesh** commands.

- Added the **Mesh → Bodies** command.

This command uses meshing methodologies which differ from those used by the **Mesh → Geometry → Surface** and **Mesh → Geometry → Solid** commands in an attempt to create a higher quality mesh by imposing fewer restrictions when meshing geometric surfaces (sheet solids), solids, and general bodies. Can be used to create a 2-D surface mesh on a connected sheet solid (stitched body) or general body or a 3-D solid tetrahedral mesh in a solid part.

> **Note:**
>
> This meshing technology has the ability to ignore internal curves and other mesh sizing specified on geometry. In order to maintain mesh sizing specified on geometry, see the options in the **Geometry** tab section below.

The geometric entities can be imported or built directly inside of Femap. When using this command and only one sheet solid, general body, or solid exists in the model, Femap automatically selects that geometric entity. If multiple sheet solids, general bodies, or solids exist, select any number of geometric entities to mesh using the standard entity selection dialog box.

> **Note:**
>
> This command can also be used to mesh geometric entities which have any number of surfaces defined using a faceted representation (convergent model) instead of a boundary representation (B-rep) or contain a mixture of both. Geometric entities containing faceted surfaces cannot be created within Femap, they can only be imported.

Once the geometric entities have been selected, the Automesh Bodies dialog box will be displayed:
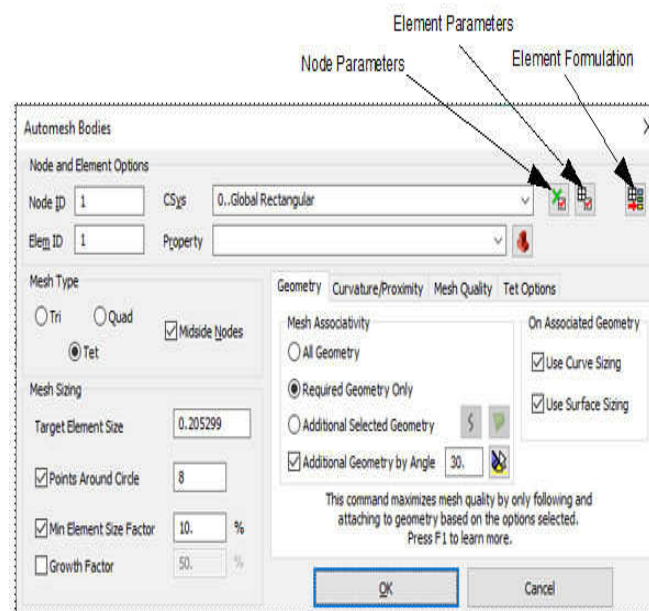
Femap will compute a **Target Element Size** to use for the selected geometric entities based upon the average curve length of all curves on the selected geometric entities. This **Target Element Size** is used in conjunction with the other parameters in the **Mesh Sizing** section, along with the settings on the **Geometry**, **Curvature/Proximity**, and **Mesh Quality** tabs to create either a triangular or quadrilateral surface mesh. When creating a tetrahedral mesh on solids, these parameters and settings are used in the same manner to create a triangular surface mesh, which is then sent to the tetrahedral mesher using options specified on the **Tet Options** tab to create the tetrahedral mesh.

If the tetrahedral mesher fails, Femap will ask "OK to Update Selector and Data Table with # nodes causing errors?" If selected answer is Yes, the problem nodes will be sent to the selection list and **Data Table** for easy identification using the **Show When Selected** tool in the **Model Info** tree and/or **Data Table**.

Node and Element Options

Most of the node and element options are explained in *Section 5.1.3.2, "Mesh, Geometry, Curve..."* in the Femap Commands guide, while other common features of this dialog box are explained in *Section 5.1.3, "Mesh, Geometry"*, also in the Femap Commands guide. When **Mesh Type** is set to **Tri** or **Quad**, an existing property from the **Plane Elements** section of the Element/Property Type dialog box, other than **Axisymmetric Shell**, must be selected from the **Property** drop-down or created using the **New Prop...** icon button. When **Mesh Type** is set to **Tet**, an existing solid property must be selected from the **Property** drop-down or new solid property created using the **New Prop...** icon button. In addition, icon buttons to specify **Node Parameters**, **Element Parameters**, and **Element Formulation** are also available (see *Section 4.2.2, "Model, Element..."* in the Femap Commands guide for more information on element formulations).

Mesh Type

These options control what type of elements will be created. Each of these options are described below.

Tri or Quad

The **Tri** option creates a surface mesh containing only triangular elements on surfaces of selected geometric entities, while the **Quad** option creates a *quadrilateral-dominant* mesh on surfaces of selected geometric entities.

> Note:
>
> The **Quad** option will attempt to create a quadrilateral-only surface mesh, but in complex geometry it is quite possible that a limited number of triangular elements will also appear in the mesh.

Tet

The **Tet** option creates a solid mesh containing only tetrahedral elements in the selected solids with each solid being meshed individually.

Midside Nodes

When activated, midside nodes are added to the elements created by this command, thus creating parabolic elements. For **Tri** and **Quad**, this option is deactivated by default, while for **Tet** this option is activated by default.

Mesh Sizing

These options control the size of the elements being created. Each of these options are described below.

Target Element Size

This value specifies the edge length that the mesher aims for in the absence of any mesh refinement. The average length of all edges in the selected geometric entities is divided by 6 to

determine the default **Target Element Size**. Decreasing the target surface size generates more refined and detailed surface mesh.

Points Around Circle

Specifies the number of triangles (edges/faces) used to define a 360 degree cylindrical surface. Any other types of cylindrical surfaces use an equivalent number of triangles. For example, a half circle uses half of this value and an ellipse would use more faces on tighter curvature.

Min Element Size Factor

When any setting causes mesh refinement to occur, the element face edges do not go below the specified value.

Growth Factor

This control determines the maximum rate at which face edge sizes vary from one face to its neighbor. The effect is only seen when there is transition from an area of refinement to the rest of the surface mesh. As an example, the default value of 50% means that a face edge can be 1.5 times its neighbor length when growing from a short to a long edge length. The recommended range for **Growth Factor** is from 10% to 60%.

Geometry tab

These options control what type of elements will be created. Each of these options are described below.

Mesh Associativity

Three options are available for **Mesh Associativity**, each of which offers some benefits and drawbacks.

All Geometry

Associates the mesh to geometry using all of the curves and surfaces of the selected geometric body. Essentially, this option provides the same level of geometric associativity as the commands on the **Mesh → Geometry...** menu, but in doing so will likely cause the mesher to create a lower quality mesh compared to what it might have been able to do if the mesh only needs to be associated to the *Required Geometry*, as the mesher does not have as much freedom to improve mesh quality by ignoring internal curves and surface.

Required Geometry Only

Associates the mesh to only the geometry which is required to be associated, such as curves and surfaces used to define geometry-based Loads, Constraints, and/or Regions. This is the default option and offers the mesher the most freedom to ignore internal curves and surfaces, which typically allows the mesher to create the highest quality mesh based on the **Target Element Size** and other specified option.

Additional Selected Geometry

Uses the same approach as **Required Geometry Only**, but allows selection of any number of curves and/or surfaces where mesh should also be associated to geometry. Additional curves are selected by clicking the **Curves** icon button and additional surfaces by clicking the **Surfaces** icon button.

> **Note:**
>
> How much the mesh quality is potentially diminished by using the **Additional Selected Geometry** option depends on the location and number of additional curves and/or surfaces selected.

When using either **Required Geometry Only** or **Additional Selected Geometry**, the **Additional Geometry by Angle** option becomes available. When enabled, automatically adds additional geometry to consider for mesh associativity based on a specified *break angle*. Default value is 30 degrees.

> **Note:**
>
> To display which additional geometry will be considered using the specified angle, click the **Show When Selected** icon button, which highlights the additional geometry in the graphics window using the current settings for the **Window → Show Entities** command. For more information, See *Section 6.3.2.3, "Window, Show Entities..."* in the Femap Commands guide.

On Associated Geometry

When **Use Curve Sizing** is activated, any mesh size currently specified for a curve on a selected geometric entity will be used during meshing. When **Use Surface Sizing** is specified, any mesh size specified for a surface on a selected geometric entity will be used during meshing. When deactivated, the mesh size on either type of entity is not considered by the mesher.

Curvature/Proximity tab

These options control settings to automatically refine mesh based on curvature and proximity. Each of these options are described below.

Surface Curvature

When on, the **Curvature Alignment** option will attempt to create a mesh where face edges follow the curvature of rounded features, such as fillets, essentially creating an *aligned* or *mapped* mesh.

The following conditions apply to the *Curvature Alignment* option:

• Aligned sections must have 4 sides and have geometric features as closed boundaries, for example:
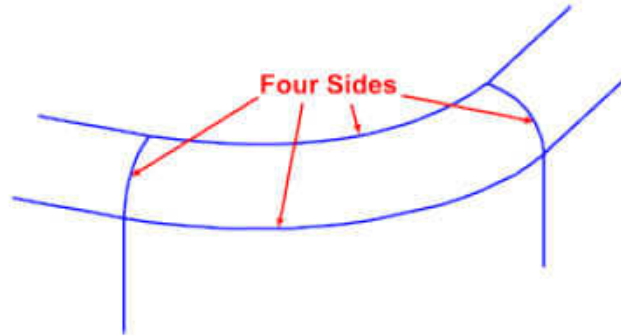
- There must be less than a 30% difference between the number of nodes created on opposing sides of an aligned section.

- The surface must have a relatively uniform surface mesh size. As a general guideline, the minimum surface size (without aligned meshing) must be greater than half of the maximum surface size.

- Aligned meshes cannot lie on a periodic interface boundary, or be connected to one along an edge (in-place interfaces are supported).

- Aligned sections cannot contain inner loops, such as an inlet duct on a filleted edge.

When **Curvature Refinement** is activated, the **Refinement Deviation Limit** is used to calculate an appropriate number of edges that are used around a complete circle. This means that the diameter of a curve is accounted for in the mesh distribution. In addition, the **Max Points Around Circle** option can be used to limit the maximum number of points around a complete circle, regardless of the value specified for **Refinement Deviation Limit**.

Once the part has been properly subdivided, the **Size on Solid** command handles all sizing automatically. Simply specify a nominal size. If further mesh grading is needed or to modify the sizes that **Size on Solid** has created - great care must be used. If the mesh size along a curve is changed manually, the mesh sizes must also be changed on all other curves in the solids needed to maintain mapped meshable surfaces. If this is not done, Femap will not be able to hex mesh the solids. While using this process, be sure to pick ALL of the required curves. In particular, at the common boundaries between sliced solids, there will be duplicate curves - one on each solid. All of these curves must get the same mesh size to maintain mesh connectivity from one solid to another.

Surface Curvature

When **Proximity Refinement** is activated, the **Elements Across** option determines the local face size by dividing the distance from one face to another across a gap by this value. It is used in conjunction with the **Max Search Distance** value, which specifies a *Search Ceiling*, and the setting for **Search Direction**, which is the direction to search with respect to face normal direction.

Max Search Distance specifies the maximum distance that the mesher includes in surface proximity, which prevents unwanted refinement. This property is useful if a component has a variable thickness and you need an option to locally refine only those areas where the thickness is

less than a specified value. However, if the gap distance is more than the this value, there is no refinement.

Search Direction can be set to **Inside**, which searches in the opposite direction of the face normal, **Outside**, which search in the same direction of the face normal, or **Both**, which searches in both directions.

Mesh/Quality tab

These options use element quality parameters to and more precise faceting to attempt to improve the overal quality of the mesh. Each of these options are described below.

Mesh Quality

The **Minimum Tri/Quad Quality** option represents a value that ranges from 0.0 to 1.0, with 0.0 being the worst and 1.0 being perfect. The quality of a triangle is calculated as the ratio of the triangle face area to the area of an equilateral triangle that would exactly fit inside the *circumcircle* of the triangle, where the circumcircle is the circle which touches all three corner points of the original face. In the diagram below, the **Minimum Face Quality** is defined as:

Minimum Face Quality = A/Aequil
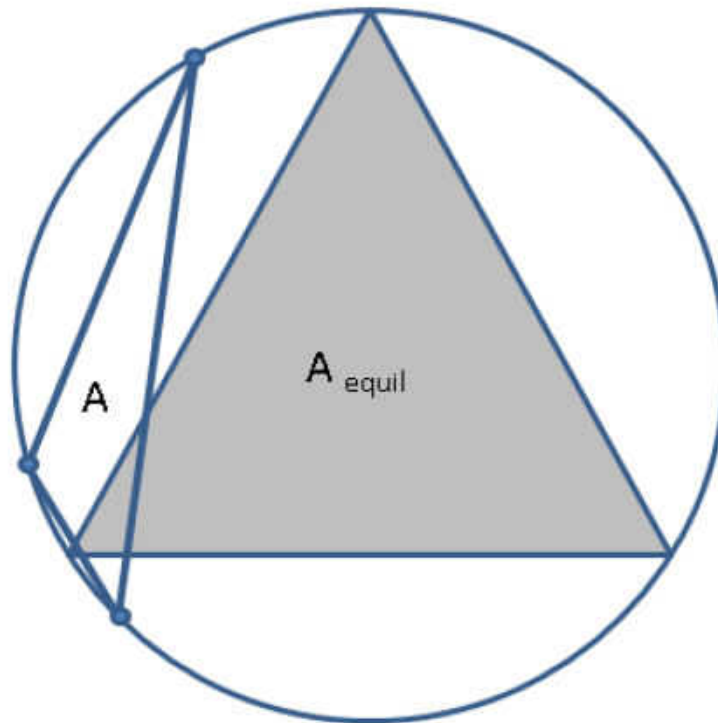
This quality check insures all the triangles in the resultant surface mesh have a quality equal to or greater than the specified value. This minimum quality is achieved by allowing the mesher to disregard feature edges, move feature vertices, or avoid projecting vertices if necessary. As a result,

the faces whose quality is below the specified value are removed or improved. Recommended values are in the range 0.01 to 0.1 and values cannot exceed 1.0.

The **Enhanced Quality Mesh** option generates triangle surfaces of higher quality. When activated, it provides the following advantages:

- Alignment with curvature and boundaries (feature edges, part boundaries). Enhanced quality triangles are better at capturing the geometry and are closer overall to the input geometry.

> Note:
>
> Alignment is only feasible when sizes do not change on a surface. In cases where sizes do change, the alignment must be sacrificed to allow for the change in mesh size.

- Allows you to better capture target sizes coming from inputs such as curvature size, surface proximity and base sizes. However, the improved quality of the triangles increases mesh generation time. You are advised to only use this option as a local control for surfaces where quality and alignment are most needed.
  When activated, the **Compatibility Refinement** option imposes a surface growth rate of 2 to limit the difference in face sizes across a gap. Therefore, if the gap between faces is small, the relative size of faces on either side of the gap is close to one another. This option improves volume mesh quality, as face sizes on opposing sides are similar without explicitly stating a size.

> Note:
>
> This option is similar to **Proximity Refinement**, but **Proximity Refinement** specifies a mesh size that is based on the gap size and number of points inside the gap. **Compatibility Refinement** ensures that the mesh size on either side of the gap is similar. In some cases, this results in fewer elements.

Faceting Tolerances

The values in this section are used to create a faceted representation of the selected geometric entities. The facets are passed to the mesher one geometric entity at a time and used to create either a triangular or quadrilateral-dominant surface mesh.

Max Facet Size facets the geometry using the specified value as a percentage of **Target Element Size** (i.e., value of 50 uses 0.5***Target Element Size** for faceting). **Distance Error** specifies the max distance from an element to a facet as a percentage of **Target Element Size**.

Angle Error is a measure of the angle between the tangent of each facet at the point it touches the edge of the solid and the chord of each facet. **Chord Error** is a measure of the length of the chord of each facet to the distance it is away from the edge of the solid at the middle of the chord. **Curve Factor** is used to improve the curve accuracy of attached curves to the surfaces. Essentially, Femap takes the values specified for solid faceting and changes the curve accuracy to be the **Curve Factor** value times more accurate.

When **Use Faceter** is activated, the current facets used to display the model in the graphics window are used by the mesher. When deactivated, the body is converted to a faceted body by a Parasolid command and those facets are used by the mesher.

> Note:
>
> If the mesher does not work as expected with **Use Faceter** activated, deactivate option and try again. A different batch of triangles are created which may allow the mesher to work

Tet Options tab

These options are used to specify options which control the creation of tetrahedral elements.

Tet Sliver Removal

When enabled, which is the default, attempts to remove tetrahedral elements considered 'slivers' near the boundary. A sliver is defined as a tetrahedral with at least one angle less than 5 degrees. In most cases this will improve mesh quality and should only be disabled if there is a problem.

Allow Void Regions

Allows meshing solids which have internal voids.

Multiple Tet thru Thickness

When activated, the tetrahedral mesher will attempt to place the specified number of elements through the thickness, from 2 to 10, throughout the model.

Tet Optimization

Sets an optimization level (mesh quality vs. speed) for the tetrahedral mesher. Using a higher value does not guarantee improved mesh quality. **3..Default** is the best compromise between speed and quality, while choosing **0..None** will simply not use any optimization. Setting to **1..Minimal** will potentially create a lower quality mesh in less time, while **5..Maximum** will potentially create a higher quality mesh, but take longer.

Tet Growth Ratio

The **Tet Growth Ratio** can modify the size of the tetrahedral elements based upon the size of elements in the surface mesh. If the **Tet Growth Ratio** value is set to 1, the size of tetrahedral elements formed will be approximately the same as the surface mesh elements. If this value is set below 1 (minimum of 0.5), the length of the tetrahedral elements formed by the solid mesher will be smaller than the surface mesh elements. If this value is above 1 (maximum of 100), the tetrahedral elements will be larger than the surface elements. If the desired tetrahedral elements need to be larger or smaller than this ratio will allow, remesh the surfaces of the solid with a finer or coarser mesh.

Mesh on Mesh

- Added the **Mesh → Mesh on Mesh** command.

...uses meshing methodologies as the **Mesh → Bodies** command (see *Section 5.1.4, "Mesh, Bodies"* in the Femap Commands guide), only it uses existing triangular or quadrilateral elements as the starting point instead of geometric entities. This is made possible by sending the facets used to display the elements in the graphics window into the mesher, which then creates a triangular or quadrilateral-dominant mesh based on those facets.

One other difference is this command cannot be used to create Tetrahedral elements, therefore the **Tet** option in the **Mesh Type** section is not available and either is the **Tet Options** tab.

The elements can be imported or built directly inside of Femap. Once the elements exist in Femap, they are selected using the standard entity selection dialog box

> Note:
>
> It may be helpful to use the **By Faces** option on the **Pick^** menu of the standard entity selection dialog box to select sections of shell elements.

Once the geometric entities have been selected, the Automesh Bodies dialog box will be displayed:
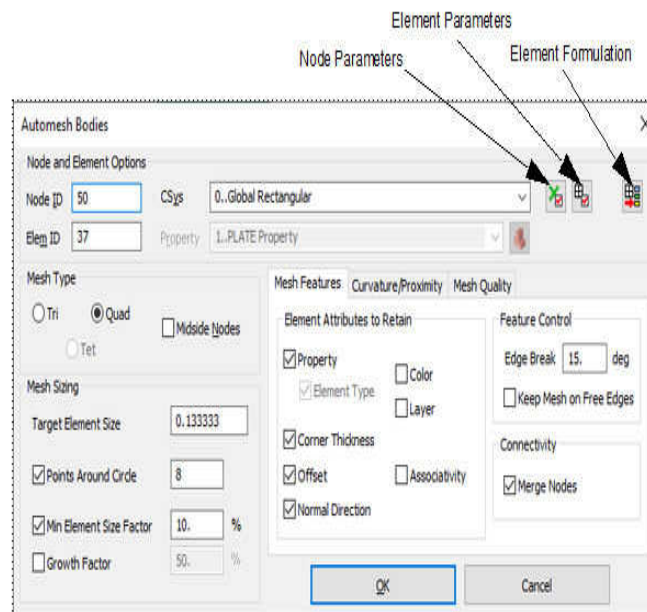
Femap will compute a **Target Element Size** to use for the selected elements based upon the average edge length of the selected elements. This **Target Element Size** is used in conjunction with the other parameters in the **Mesh Sizing** section, along with the settings on the **Mesh Features, Curvature/Proximity,** and **Mesh Quality** tabs to create either a triangular or quadrilateral-dominant mesh which will replace an existing mesh.

Node and Element Options

Most of the node and element options are explained in *Section 5.1.3.2, "Mesh, Geometry, Curve..."* in the Femap Commands guide, while other common features of this dialog box are explained in *Section 5.1.3, "Mesh, Geometry"*, also in the Femap Commands guide.

When **Mesh Type** is set to **Tri** or **Quad**, the original property of each element will be retained by default. To select a different property from the **Property** drop-down to use for the newly created elements, the **Property** option in the **Element Attributes to Retain** section of the **Mesh Features** tab must be deactivated. Once deactivated, an existing property from the **Plane Elements** section of Element/Property Type dialog box, other than **Axisymmetric Shell**, may be selected from the **Property** drop-down or created using the **New Prop...** icon button.

Mesh Features tab

These options control which element attributes will be retained along with some additional options.

Mesh Attributes to Retain

This section contains a number of options which control which element attributes will be retained once the current elements have been replaced with the new elements created by the command.

Property

When activated, the property of the original element is retained.

Element Type

When activated, the element type of the original element is retained and only elements which were originally an element type which matches the property type of the selected property will be remeshed.

Corner Thickness

When activated, the corner thickness values specified for each original element will be retained.

Offset

When activated, the element offset value for each original element will be retained.

Normal Direction

When activated, the normal direction of each original element will be retained.

Color

When activated, the element color of each original element will be retained.

Layer

When activated, the layer of each original element will be retained.

Associativity

When activated, the geometry associativity for each original element will be retained.

Feature Control

When selecting elements for this command, it may be easier to select elements which span multiple *mesh regions* that are adjacent to one another. Mesh regions can be automatically determined using an angle, specified in degrees, for **Edge Break**. Femap calculates the normal to the planar face for each element. If the normals of two adjacent elements are within the angle that you specify of being parallel, they will be considered to be in the same segment. If they are not within that angle, multiple segments will be formed.

When the **Keep Mesh on Free Edges** option is activated, the nodes which lie on free edges of the selected elements will be retained. Otherwise, the mesh along the free edges can be modified by the mesher.

Connectivity

When the **Merge Nodes** option is activated, any new nodes created by the command will be merged with other new nodes and existing nodes when the nodes fall within the default **Merge Tolerance**. For more information about the default **Merge Tolerance**, see *Section 7.4.1, "Tools, Parameters..."* in the Femap Commands guide.

Elements

- Added ability to create a CBUSH1D element for Nastran solvers by using a Spring/Damper element which references a Spring/Damper property with the CBUSH1D option selected or by using a Spring/Damper to Ground element which references a Spring/Damper to Ground property with the CBUSH1D option selected.

Properties

- Added a **CBUSH1D** option to both the Spring/Damper property and the Spring Damper to Ground property types. When a Spring/Damper element or Spring/Damper to Ground element references a property which is set to **CBUSH1D**, a CBUSH1D and corresponding PBUSH1D will be written to the Nastran input file. In addition, the Nonlinear/Freq Resp... button can be used to access the Define Nonlinear 1D Spring/Damper dialog box to choose Functions or Design Equations to control Spring, Damper, Shock Absorber, and/or Generalized Spring/Damper behavior on the PBUSH1D entry.

Simulation Entities

- Added the **Model → Simulation Entities → Design Equation** command, which opens the NASTRAN Design Equation Manager to manage design equation entities in a model.
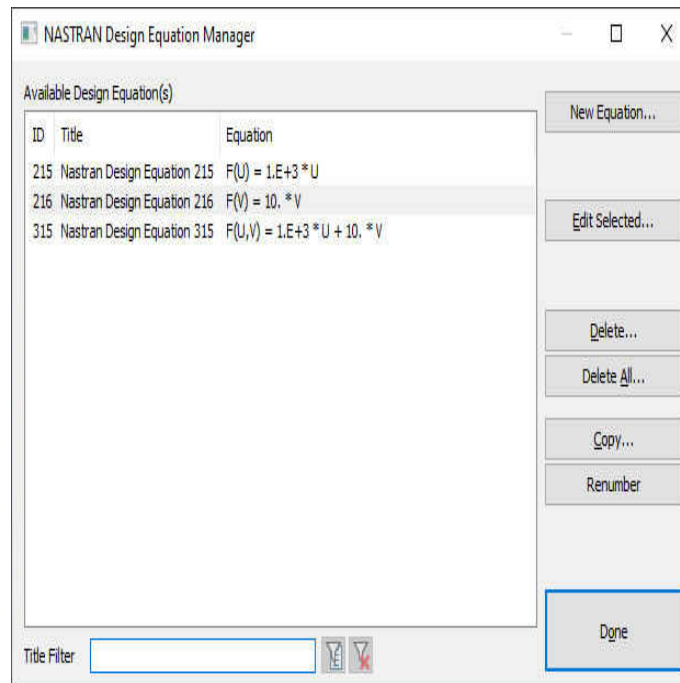
This manager is used to create Design Equation entities (DEQATNs) for Nastran solvers.

This manager can be used to create a **New Equation**; **Edit Selected** (equations), **Delete**, **Copy**, or **Renumber** Design Equation(s) currently highlighted in the **Available Design Equation(s)** list; or **Delete All** the Design Equations currently in the model.

Clicking the **New Equation** button opens the NASTRAN Design Equation dialog box:
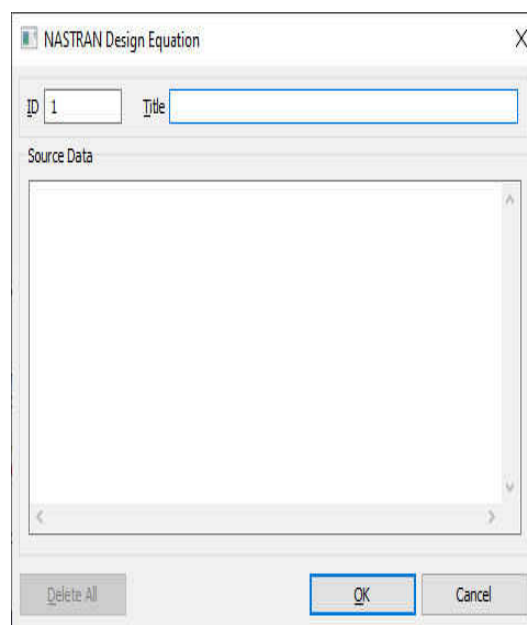
Specify an **ID** and **Title**, then enter the text of the Design Equation (i.e., the text formatted exactly as it should appear in the EQUATION field of the DEQATN entry) into the **Source Data** section. To delete all of the text currently in the **Source Data** section, us the **Delete All** button.

> Note:
>
> When Design Equations are imported into Femap from DEQATN entries in a Nastran input file, the exact format of the text from the EQUATION field of the DEQATN entry is maintained in the **Source Data** section.

In Femap, there is currently only one use for Design Equations, which is to allow selection of a Design Equation using drop-downs in various sections of the Define Nonlinear 1D Spring/Damper when that section is set to Equation. For more information, see "Spring/Damper Element Properties" in *Section 4.2.2.1, "Line Elements"* in the Femap Commands guide.

Groups and Layers

- Added the **Group → Element → All Nodes on Solid** command which places any element which has all its nodes associated to the selected solids in a group.

Output and Post-Processing

- Updated the **View → Advanced Post → Stress Linearization** command to copy the underlying transformed stresses to the clipboard instead of the regular output report by holding down the Shift key when pressing the **Copy to Clipboard** button.

Geometry Interfaces

The following Femap interfaces have been updated to support newer geometry formats:

Femap Interface	Latest Supported Version
Parasolid	33.1
NX	version 1953
ACIS	2020.1.0 1.0.1
Creo (Pro/Engineer)	Creo 7
SolidWorks	2021

- Added ability to import Parasolid files with solids, sheet solids, and/or general bodies which have any number of surfaces using a faceted representation (convergent modeling) instead of a boundary representation (b-rep).

For details, see *Section 9, "Geometry Interfaces"* in the Femap User Guide.

Analysis Program Interfaces

Several of the analysis program interfaces have been improved. These changes include:

- Femap Neutral File Interface

- Simcenter Nastran Interface

- Nastran Interfaces (Simcenter and MSC/MD)

- LS-Dyna Interface

For details, see *Section 8, "Analysis Program Interfaces"* in the Femap User Guide.

Femap Neutral File Interface

- Updated Neutral Read and Write for v2021.1 changes.

Simcenter Nastran Interface

- Added support to specify a Connection Property when the **Gaps as Contact** option is enabled for SOL 101.

SOL 401/402

- Added support for reading and writing the MPLAS entry used by SOL 401 and SOL 402.

A number of bugs were corrected.

For details, see *Section 8, "Analysis Program Interfaces"* in the Femap User Guide.

Nastran Interfaces (Simcenter and MSC/MD)

- Added support for defining dynamic subcases for **3..Transient Dynamic/Time History**, **4..Frequency/Har-monic Response**, and **12..Nonlinear Transient Response** analysis types. In addition, a static subcase can now be created for **3..Transient Dynamic/Time History** and **4..Frequency/Harmonic Response** for the purpose of accounting for differential stiffness using a STATSUB entry.

- Added support to read and write the CBUSH1D element type and corresponding PBUSH1D property type entries. In addition, general nonlinear behavior can be defined with Functions or Design Equations.

- Added support to read and write the DEQATN entry by using a Design Equation Simulation Entity. In addition, general nonlinear behavior can be defined with Functions or Design Equations.

- Updated the default value of the PRGPST option in the NASTRAN Bulk Data Options dialog box to off.

A number of bugs were corrected.

For details, see *Section 8, "Analysis Program Interfaces"* in the Femap User Guide.

LS-Dyna Interface

- Added support for element formulation EQ18 and EQ-18 for solid elements as well as element formulation EQ16 for shell elements.

A number of bugs were corrected.

For details, see *Section 8, "Analysis Program Interfaces"* in the Femap User Guide.

Tools

- Updated the **Tools → Check → Mesh Interference** command to have selected Plot Only plates elements be treated as having zero thickness, removing numerical roundoff and false positive mesh interferences.

JT Files

- Added the ability to output the active, visible, selected, or all load sets and/or constraint sets to a JT file when **Type** is set to **Standard Output** in the JT File Options dialog box. In addition, element symbols can now be output to a JT file.

- Added a **Multi Result Output** option to the JT File Options dialog box which allows for the selection of any number of output vectors in any number of output sets to be output to a JT file either as **Raw Output Data** or as **Collated Vectors and Tensors** with options to use averaging, transformation, and/or include corner data.

OLE/COM API

New and modified API Objects and Attributes

- Added a BodyMesher (feBodyMesher) object to the API.

Added MeshType, MidsideNodes, TargetSize, MinSizeFactorOn, MinSizeFactor, PointsAroundCircleOn, PointsA-roundCircle, GrowthFactorOn, and GrowthFactor attributes to control **Mesh Sizing** options for the **Mesh → Bodies** and **Mesh → Mesh on Mesh** commands when using the BodyMesher Object.

Added AssociateTo, AssociateAngleOn, AssociateAngle, UseCurveSizing, and UseSurfaceSizing attributes to control options on the **Geometry** tab of the **Mesh → Bodies** command when using the BodyMesher Object.

Added RetainProperty, RetainElemType, RetainCornerThickness, RetainOffset, RetainNormalDirection, Retain-Color, RetainLayer, RetainAssociativity, FeatureEdgeBreakAngle, KeepMeshOnFreeEdges, and MergeNodes attributes to control options on the **Mesh Features** tab of the **Mesh → Mesh on Mesh** command when using the Body-Mesher Object.

Added CurvatureAlignment, CurvatureRefinement, CurvatureRefinementMaxPoints, CurvatureRefinementLimit, ProximityRefinement, ProximityElemAcrossOn, ProximityElemAcross, ProximityDistanceOn, ProximityDis-tance, ProximityDirection attributes to control options on the **Curvature/Proximity** tab of the **Mesh → Bodies** and **Mesh → Mesh on Mesh** commands when using the BodyMesher Object.

Added MinQualityOn, MinQuality, EnhancedQualityMesh, CompatibilityRefinement, FacetMaxSize, FacetDistanceError, FacetAngleError, FacetChordError, FacetCurveFactor, and UseFaceter attributes to control options on the **Mesh Quality** tab of the **Mesh → Bodies** and **Mesh → Mesh on Mesh** commands when using the BodyMesher Object.

Added TetSliverRemoval, TetAllowVoids, TetMultipleThruThicknessOn, TetMultipleThruThickness, TetOptimizationLevel, and TetGrowthRatio attributes to control options on the **Tet Options** tab of the **Mesh → Bodies** command when using the BodyMesher Object.

New and Updated API Methods

- Added Clear, Dialog, Size, UseSizing, Associate, MeshBodies, MeshOnMesh, CalculateDefaultMeshSize, Set-SizeOnCurves, and SetSizeOnSurfaces methods to the BodyMesher Object.

- Added NodesAsSet method to the BCSet Object.

- Added NodesAsSet and ElementsAsSet methods to the ConnectionRegion Object.

- Added IsFacet method to the Curve Object.

- Added NodesAsSet and ElementsAsSet methods to the LoadSet Object.

- Added IsFacet and IsAllFacet methods to the Solid Object.

- Added IsFacet method to the Surface Object.

The following functions have been added or updated:

- feFilePictureSaveJT

- feFilePictureSaveJTMultiResults

- feAppAllowUVSurfaceForMeshFaces

4. What's New - version 2021.1 MP 1

Analysis Program Interfaces

Several of the analysis program interfaces have been improved. These changes include:

- **Simcenter Nastran Interface**

For details, see *Section 8, "Analysis Program Interfaces"* in the Femap User Guide.

Simcenter Nastran Interface

SOL 401/402

Added support to write the CRSTRN case control entry to the input file if either the **Total Strain** or **Elastic Strain** output request is activated in the NASTRAN Output Requests dialog box.

5. What's New - Version 2021.1

User Interface

General

- Added functionality to hide the list of entity ranges in the standard selection dialog box to improve performance when more than 1000 unique ID ranges have been selected.
  Once more then 1000 "unique ID ranges" are selected, the list of selected entities is replaced by text stating **(number of unique ranges) Ranges Selected** and **Show List>>>**:
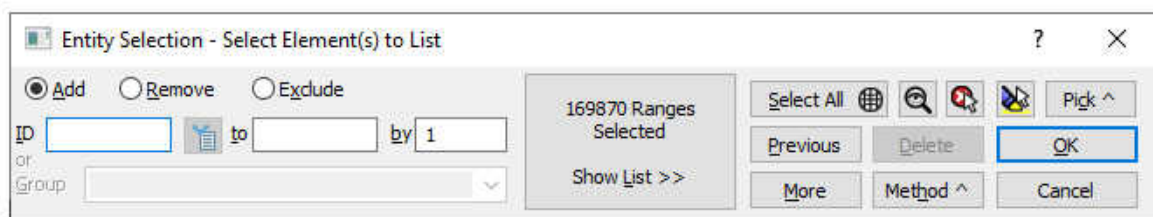
  To display the entire list of ranges, click anywhere in the list area. To hide the list again, click the **Hide List** button which appears to the left of the list area.

- Added the **Match Color** button menu to the Color Palette, which selects a color by matching the color of an existing entity.

- Improved handling of *repeated error messages* to catch cases where multiple errors were repeated in a pattern. This can dramatically improve import times of invalid or incomplete Nastran models.

- Enhanced performance of picking, especially front picking (up to 15 times faster) and area picking (up to 10 times faster), when using the **New Picking** option on the **Graphics** tab in the Preferences dialog box. **New Picking** also takes advantage of OpenMP technology.

Menu

- Added the **Tools → Other Windows → Mesh Control Explorer** command. See Mesh Control Explorer - New for 2021.1 for more information.

Meshing Toolbox

- Added an icon to open the **Mesh Control Explorer** dockable pane from within the **Meshing** Toolbox. For more information, see Mesh Control Explorer - New for 2021.1.

Function/Table Editor

- Added the **Connect To Excel** command which is only available after the **Send To Excel** command on the **Copy to Clipboard** menu has been used to send data to Excel. **Connect to Excel** opens the Excel Connection dialog box. When the **Live** option is activated, any additions and modifications done in Excel are automatically reflected in the **Function/Table Editor**. If the **Live** option is deactivated, then additions and modifications can be committed to the **Function/Table Editor** by using the **Update** button.

Mesh Control Explorer - New for 2021.1

Added the **Mesh Control Explorer** to perform a number of different tasks, including specification of options for mesh propagation, visualization of slaved surfaces for solid meshing and/or mesh approaches on surfaces, and determining if curves are paired by a mesh approach and/or can be automatically paired based on proximity when mesh sizing.

The **Mesh Control Explorer** pane contains a number of icons and a section used to list the results of the various exploration tools:
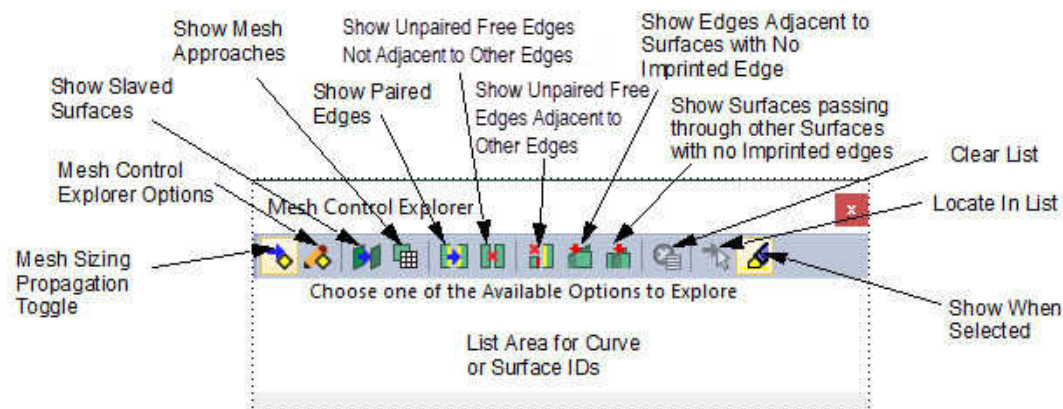
Mesh Control Explorer Icons

The icons are split into five sections. The first section is used to specify options for automatic mesh propagation.

The second section contains tools which are used to visualize how surfaces of different solids are linked to one another for solid meshing or determine which surfaces have a mesh approach currently assigned.

The third section contains tools to visualize curves on geometric entities which are already automatically *paired* and/or can be when using the mesh propagation options; adjacent to one another, but require additional modification, such as being split at various locations, to allow automatic *pairing*; or not adjacent to any other curves, therefore not eligible for automatic pairing unless existing geometry is moved or new geometry is created.

The fourth section contains a single icon to clear the list area, while the fifth section contains one command which locates the line in the list area containing the curve or surface selected from the graphics window and another which controls how curves or surfaces currently highlighted in the list area are highlighted in the graphics window.

Mesh Sizing Propagation Toggle

When activated, uses options in the Mesh Sizing Propagation Options dialog box to set up mesh sizing on curves to facilitate mesh propagation from geometric entity-to-geometric entity. To access the Mesh Sizing Propagation Options dialog box via the **Mesh Control Explorer**, click the **Mesh Control Explorer Options** icon. When deactivated, the options in the Mesh Sizing Propagation Options dialog box are ignored.

Mesh Control Explorer Options

Opens the Mesh Sizing Propagation Options dialog box. For more information, see **Meshing**.

Show Slaved Surfaces

This tool is used to visualize how surfaces of solids are linked to one another using the "Matched - Link to Surface" **Mesh Approach**. When a surface is "linked" to another surface, the Slave surface is given an identical mesh to the master surface, which allows either the hex mesher or tetrahedral mesher to create a single connected mesh across multiple solids.

When this tool is used, a list of all surfaces which are linked using "Matched - Link to Surface" **Mesh Approach** will always be sent to the list area. In addition, when highlighting is enabled, which is the default, all surfaces which have the mesh approach defined or are referenced will also be highlighted in the graphics window. Each master will appear "filled", while each Slave will appear "outlined". Once the list area is populated, any single line can be selected, which highlights only the surfaces on that line in the graphics area.

Each line in the list area can contain any combination of Surface IDs and symbols, with the location of the Surface ID and type of symbol holding particular significance. In all cases, the surface ID to the left of the symbol is the master, while the surface ID to the right of the symbol is the Slave.

If a "<" symbol is between two Surface IDs, this indicates the surfaces reside on the same solid. In general, this indicates the solid has been sized for hex meshing and the hex mesh will start at the master and continue to the Slave surface, following all the curves found on the solid between the two surfaces.

If a "<<" symbol is between two Surface IDs, this indicates the surfaces reside on different solids. In general, these surfaces have been linked specifically so the Slave surface will be meshed identically to the master, which allows for the mesh on both solids to be connected at that interface (i.e., share nodes at that surface). In Femap, this type of linking is often referred to as adjacent surface matching.

It is quite common for a surface to be the master or Slave for hex meshing purposes on a particular solid, but then be the opposite when setting up adjacent surface matching. For instance, surfaces A, B, C, and D all appear on a single line in the list area as "A<<B<C<<D". This means:

- A is any surface on "Solid 1" (could potentially be matched to another surface for hex meshing purposes)

- B and C are both on "Solid 2", where B is the "master" and C is the "slave" for hex meshing purposes.

- D is any surface on "Solid 3" (could potentially be matched to another surface for hex meshing purposes).

- In the connection between "Solid 1" and "Solid 2", A is the "master" and B is the "slave".

- In the connection between "Solid 2 and "Solid 3", C is the "master" and D is the "slave".

Right mouse-clicking on any selected line in the list area will display a context-sensitive menu containing four commands.

Reverse reverses the master-Slave relationships for all surfaces on the selected line. If all relationships cannot be reversed, an error stating "Unable to **Reverse**. Multiple Surfaces are Slaved to Surface (ID)." will be displayed.

Approach on Master will display the Surface Mesh Approach dialog box, allowing the definition of a meshing approach on the master. For more information on the available mesh approaches, see *Section 5.1.2.15, "Mesh, Mesh Control, Approach On Surface"* in the Femap Commands guide.

List simply lists all the surfaces on the selected line to the **Messages** window, while **Show All** highlights all of the surfaces found by the tool using the current options specified on the **Show When Selected** icon menu. These commands are also found on the context-sensitive menu for every tool in the **Mesh Control Explorer** which has a con-text-sensitive menu, only they list or highlight all of the found curves instead surfaces.

For example, the geometry below is sized for hex meshing. The surface IDs on each line in the list area of the **Mesh Control Explorer** each represent any number of pairs of surfaces which are "linked" using the "Matched - Link to Surface" **Mesh Approach**. In addition, those surfaces are all simultaneously highlighted in the graphics window.
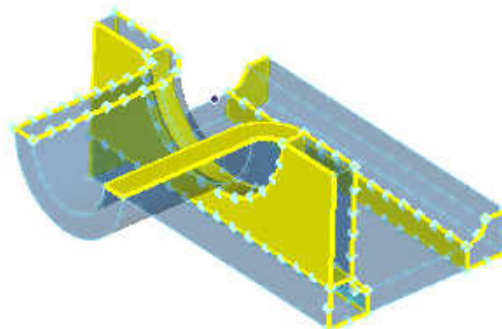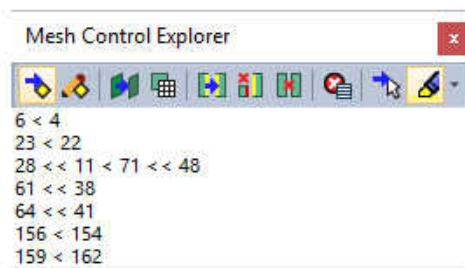
Show Mesh Approaches

This tool determines which surfaces have a mesh approach assigned (other than **Matched - Linked to Surface**).

When this tool is used, a list of all surfaces which have a mesh approach assigned will always be sent to the list area. In addition, when highlighting is enabled, which is the default, all surfaces which have a mesh approach assigned will also be highlighted in the graphics window. Once the list area is populated, any single line can be selected, which highlights only the surface on that line in the graphics area.

Right mouse-clicking on any selected line in the list area will display a context-sensitive menu containing three commands.

Approach displays the Surface Mesh Approach dialog box, which can be used to review or edit the mesh approach currently assigned to the surface. For more information on the available mesh approaches, see *Section 5.1.2.15, "Mesh, Mesh Control, Approach On Surface"* in the Femap Commands guide.

List simply lists all the surfaces on the selected line to the Messages window, while Show All highlights all of the surfaces found by the tool using the current options specified on the **Show When Selected** icon menu.

Show Paired Edges

This tool is used to determine which curves have been "paired" in some way due to mesh approach and/or can be automatically "paired" when **Mesh Sizing Propagation** is activated because the curves are coincident and share end points, then lists them to the list area. When **Highlighting** is activated, which is the default, all curves which are paired will also be highlighted in the graphics window. When a solid is sized for hex meshing, the curves which extend from the master surface to the Slave surface are "paired" because all of them need to have the same mesh size in order for mesh properly extrude, revolve, and/or sweep through the solid to create "layers" of solid hex (brick) or wedge elements.

When "paired", curve IDs will be listed between brackets, separated by commas. For example:

- Two paired curves, "A" and "B", are on two different solids, sheet solids, and/or general bodies, this listing is "[Curve A ID] , [Curve B ID]"

- Four paired curves, "A","B", "C", and "D", are on the same solid the listing is "[Curve A ID, Curve B ID, Curve C ID, Curve D ID]".

- Two paired curves, "A" and "B", are on "Solid 1", while two paired curves, "C" and "D", are on "Solid 2". If properly set up for solid meshing (i.e., curves "B" and "C" are on surfaces which have a "Matched - Linked to Surface" mesh approach defined), the listing is "[Curve A ID, Curve B ID] , [Curve C ID, Curve D ID]".

Right-mouse clicking on any selected line in the list area displays a context-sensitive menu. Use List to simply lists all the curves on the selected line to the Messages window, while **Show All** highlights all of the curves found by the tool using the current options specified on the **Show When Selected** icon menu.

Show Unpaired Free Edges that are Not Adjacent to Other Edges

This tool is used to visualize curves which are not coincident to other curves and also do not share any end points. Many times this tool will simply highlight geometric free edges in the model. When highlighting is activated, which is the default, all curves which are found by the tool will also be highlighted in the graphics window. Curves found by this tool are always listed individually.

List simply lists the curve on the selected line to the **Messages** window, while **Show All** highlights all of the curves found by the tool using the current options specified on the **Show When Selected** icon menu.

Show Unpaired Free Edges that are Adjacent to Other Edges

This tool is used to determine curves which are partially coincident to other curves, but do not share any end points, then lists them to the list area. When highlighting is activated, which is the default, all curves which are found by the tool will also be highlighted in the graphics window. These curves are listed as "[Curve A ID] , [Curve B ID]".

To potentially create shared end points on a particular set of unpaired curves, select a line in the list area, then right-mouse click and choose the **Imprint Points** command from the context-sensitive menu. To attempt to create shared end points on all of the unpaired curves currently in the list area, select any line from the list area, then right-mouse click and choose the **Imprint Points - All** command from the context-sensitive menu.

List simply lists the curve on the selected line to the **Messages** window, while **Show All** highlights all of the curves found by the tool using the current options specified on the **Show When Selected** icon menu.

Show Edges that are Adjacent to Surfaces with No Imprinted Edges

This tool is used to determine if edges of any number of surfaces are adjacent to the face of a surface, but there is no imprinted edge where an edge(s) of the surface(s) meet the face of another surface (T-Junctions) then lists them to the list area. When highlighting is enabled, which is the default, all *Surface*

Face-to-Surface Edge combinations which are found by the tool will also be highlighted in the graphics window. For a single combination, this is listed as "Surface ID > [Surface Edge (Curve) ID]". For combinations where edges from multiple surface intersect a single surface, this is listed as "Surface ID > [Surface Edge (Curve) ID A, Surface Edge (Curve) ID B, etc]".

To potentially create imprinted edges on the surface using the surface edges, select a line in the list area, then right-mouse click and choose the **Imprint Edges** from the context-sensitive menu. To attempt to create imprinted edges on all of the *Surface Face-to-Surface Edge* combinations currently in the list area, select any line from the list area, then right-mouse click and choose the **Imprint Edges - All** command from the context-sensitive menu. In addition, if you would like to have the imprinted edges always be extended to the extents of the surface in both directions, activate the **Extend Imprints** toggle on the context-sensitive menu.

List simply lists the surface ID and all curve IDs on the selected line to the **Messages** window, while **Show All** highlights found by the tool using the current options specified on the **Show When Selected** icon menu.

Show Surfaces that Pierce Other Surfaces with No Imprinted Intersection

This tool is used to determine if any number of surfaces pierce through another surface but there are no imprinted curves at the location where the surfaces intersect., then lists them to the list area. When highlighting is enabled, which is the default, all *Surface(s) piercing another Surface without Imprinted Intersection(s)* combinations which are found by the tool will also be highlighted in the graphics window. For a single combination, this is listed as "Surface being pierced ID > [Piercing Surface ID]". For combinations where multiple surfaces are piercing a single surface, this is listed as "Surface being pierced ID > [Piercing Surface ID A, Piercing Surface ID B, etc}".

To potentially create imprinted edges at the intersections of surface(s) which pierce each other, select a line in the list area, then right-mouse click and choose the **Imprint Intersections** from the context-sensitive menu. To attempt to create imprinted edges on all of the surfaces which pierce each other currently in the list area, select any line from the list area, then right-mouse click and choose the **Imprint Intersections - All** command from the context-sensitive menu. In addition, if you would like to have the imprinted edges always be extended to the extents of the surface(s) in both directions, enable the **Extend Imprints** toggle on the context-sensitive menu.
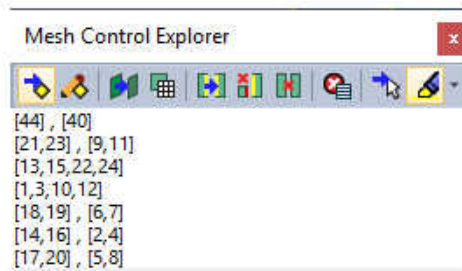
List simply lists the curve on the selected line to the **Messages** window, while **Show All** highlights all of the curves found by the tool using the current options specified on the **Show When Selected** icon menu.

Clear List

This icon is used to clear all lines of curve IDs or surface IDs from the list area.

Locate in List

This icon is only available when the list area is populated with curve or surface IDs by one of the tools. When curve IDs are listed, this icon allows selection of a curve from the graphics window, then locates and highlights the curve in the list area. When surface IDs are listed, this icon locates and highlights the selected surface in this list area.

Show When Selected menu

Contains a number of options to show the entities currently highlighted in the **Mesh Control Explorer** in the main graphics window. Only Curves and Surfaces can be "shown" and which type of entity being shown depends on which tool was used to populate the list in the **Mesh Control Explorer**. By default, this command is enabled. The commands on this menu use the various options of the **Window → Show Entities** command (see *Section 6.3.2.3, "Window, Show Entities..."* in the Femap Commands guide). Once the options have been selected, simply clicking the **Show When Selected** icon will enable/disable the highlighting.

When highlighting is enabled, all entities which are listed to the **Mesh Control Explorer** by tool will be highlighted in the graphics window. If a single line is then highlighted, only the entities on that line will be highlighted.

Setting the menu to **Highlight** highlights the entities in the graphics window. **Transparent Highlight** does the same thing as **Highlight**, but will make all non-selected entities temporarily transparent as well.

> Note:
>
> The level of transparency used in the **Transparent Highlight** option can be adjusted using a global value for all entities using **View → Options**; Category: **Tools and View Style**; **Transparency** option.

Setting the menu to **Show Selected Only** will cause all entities NOT associated with the selected entities in the **Mesh Control Explorer** to temporarily disappear until **Show When Selected** is toggled off or the model is regenerated. The **Autoscale to Show Entities**, **Show Labels**, and **Show Normals** (shows element normals) options can be toggled and the highlight color can also be chosen from the Femap color palette using the **Highlight Color...** command.

Geometry

- Added an **Overlap Repair** option to the Automatic Midsurface Generation dialog boxwhich is accessed by the **Geometry → Midsurface → Automatic** command.
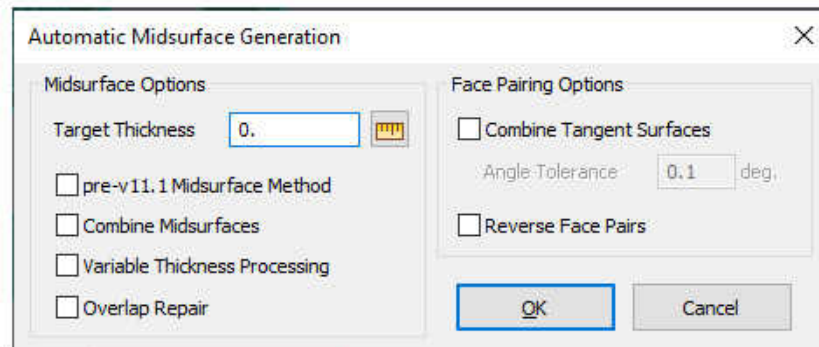
When activated, **Overlap Repair** attempts to automatically remove overlapping midsurfaces caused by offsets in the original solid geometry, thereby resulting in fewer newly created midsurfaces and/or midsurfaces which do not overlap. This option may be useful when creating midsurfaces in cases where multiple internal ribs approach an "intersection" or "near intersection" from various directions and/or the ribs on "opposite sides" of the "intersection" are otherwise not aligned with one another.

- Added **Enhanced Geometry → Curve - Centerline** to better handle more complex beam-like solids, including helical and swept shapes created from complex splines.

Meshing

- Added a **Size Propagation** section to the dialog box for the **Mesh → Mesh Control → Size on Curve**; **Mesh → Mesh Control → Size on Surface**; and **Mesh → Mesh Control → Size on Solid** commands. In each command, there is a **Propagate Sizing** option and an icon button which accesses the Mesh Sizing Propagation Options dialog box. The **Propagate Sizing** option can also be activated and the other options specified via the **Mesh Control Explorer** dockable pane, which was added for 2021.1. Opens the Mesh Sizing Propagation Options dialog box:
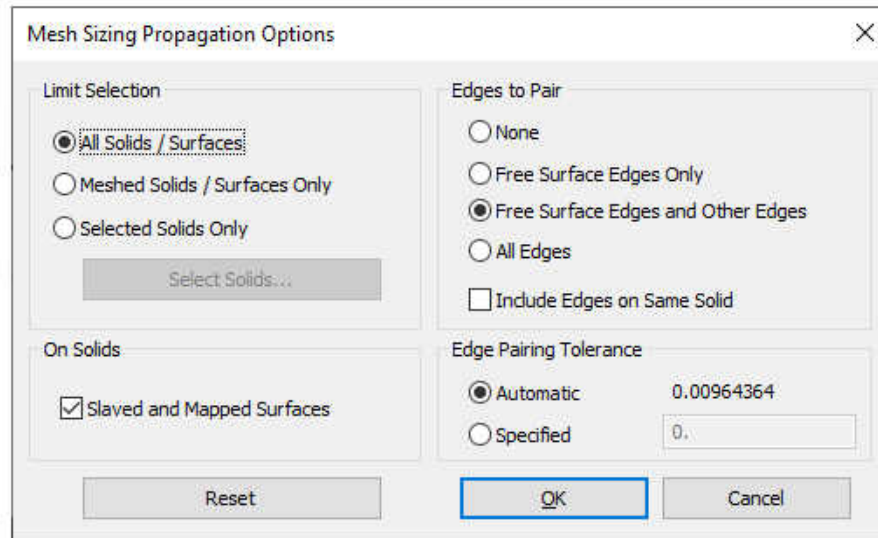
This dialog box is broken into four sections, each controlling a different aspect of mesh sizing propagation.

Limit Selection

Controls which solids/surfaces are considered when mesh sizes are updated. **Choose All Solids/ Surfaces**, **Meshed Solids/Surfaces Only**, or **Selected Solids Only**. When using **Selected Solid Only**, click the **Select Solids** button to select solids, sheet solids, and general bodies using the standard entity selection dialog box.

On Solids

If **Slaved and Mapped** Surfaces is activated, then slaved surfaces with mapped mesh approaches are con-sidered when determining which edges to update, otherwise they are not considered.

Edges to Pair

These options will automatically find neighboring edges of unstitched geometric entities (i.e., individual surfaces, sheet solids, and/or non-enclosed volume portions of general bodies) and attempt to match mesh sizes across the multiple coincident edges.

> Note:
>
> To be identified and linked by any option in this section, edges must have coincident endpoints.

Off

No edges are paired in this manner.

Free Surface Edges Only

Only edges that are used by one surface will be considered. This is the default, however it will not identify certain types of geometry, such as T-junctions, where one of the edges is an internal edge of sheet solid or general body (i.e., surfaces which have been stitched or non-manifold

added together). On the other hand, it will identify T-junctions where all curves that meet at the junction belong to individual surfaces.

Free Surface Edges and Other Edges

Similar to **Free Surface Edges Only**, but also considers internal edges of any sheet solid or general body.

All Edges

Simply checks all geometric edges. This method will have the slowest performance, but also considers edges of regular (non-sheet) solids, which allows it to identify junctions between solids and sheet solids.

> Note:
>
> If **Include Edges on Same Solid** is deactivated, which is the default, edges are never checked for pairing with other edges of the same sheet solid, solid, or general body. For instance, if there is a sheet solid that wraps around to form a closed cylinder, the edges would never be linked unless this option is activated.

Edge Pairing Tolerance

This is the tolerance used to automatically pair coincident edges. **Automatic** uses the default node merge tolerance, while **Specified** uses value provided by the user.

> Note:
>
> Femap determines the default node merge tolerance based on overall model size. The number is 1/10000 of the *model box diagonal* (think of the model box being an invisible box that completely encapsulates every entity in the model).

- Added a **Detect Surface Poles** option to **Mapped** approaches in the Surface Mesh Approach dialog box. When activated, the location of surface poles will be detected and the selected points may be reordered to create a higher-quality mesh. When deactivated, the points are always used as entered.

Materials

- Added option **607..MSC.Nastran Cohesive Material (MCOHE SOL400)** to **Other Types** to support the MCOHE entry for MSC Nastran SOL 400.

- Added option **608..MSC.Nastran Gasket Material (MATG SOL400, SOL 600)** to **Other Types** to support the MATG entry for MSC Nastran SOL 400 and SOL 600.

Properties

- Added a **Define Property Extensions** icon button to the Define Property dialog boxes for a number of different property types. When clicked, the Property Extensions dialog box is displayed, which allows specification of solver-specific items for the current property type. The options available change from

property type-to-property type. The **Filter** icon button can be used to limit the tabs which are displayed in the Property Extensions dialog box.

Options available for Integration Type are "0..Default", "1..Gauss", or "2..Newton-Coates". Options available for Output Type are "0..Default", "1..Corner Nodes", or "2..Gauss Points". Options available for Matrix Scheme are "0..Default", "1..Secant-Type Matrix", or "2..Tangent-Type Matrix".

Property Extensions - Rod

MSC Nastran tab - These options are only used for SOL 400 and written to a PRODN1 entry. When **Large Strain** is activated, an existing material may then be selected from the **Material** drop-down in the **Overrides** section.

Property Extensions - Bar and Beam

MSC Nastran tab - These options are only used for SOL 400 and written to a PBARN1 entry for a bar or a PBEMN1 entry for a beam. When **Large Strain** is activated, the options for **Integration Type** are **0..Default**, **1..Linear/Cubic**, **2..Linear/Cubic Closed Section**, **3..Linear/Cubic Open Section**, or **4..Linear-shear**, while the options for **Section** are **0..Default**, **1..Smeared Cross-Section**, or **2..Numerically Integrated Cross Section**. In addition, an existing material may then be selected from the **Material** drop-down in the **Overrides** section.

Property Extensions - Membrane, Bending Only, and Plate

Autodesk Nastran tab - For plate properties, the **General** section can be used to specify which element stress **Component Direction** will be used to determine element failure and maximum **Compression Allowable**.

The **Shell Type After Reversion** drop-down specifies the type of element being created, **1..Tension Only** or **2..Shear Panel**.

When set to **1..Tension Only** the values defined for **X Compression Factor** and **Y Compression Factor** are used to determine the stiffness when the element fails.

When set to **2..Shear Panel**, **Effectiveness Factor**, F1, F2, F3, and F4 specify treatment of the effective extensional area of the shear panel.

MSC Nastran tab - These options are only used for SOL 400 and written to a PSHLN1 entry. When **Large Strain** is activated, two options, **Integration Type** and **Behavior Type**, are available for each element topology.

For a Membrane, Bending Only, or Plate property, available options for **Integration Type** and **Behavior Type** for each Element Topology for Plates:

Element Topology	Integration Type	Behavior Type
Tria3 Elements	0..Default	0..Default

Element Topology	Integration Type	Behavior Type
	1..Linear	2..Doubly-curved thin shell
	5..Linear Discrete Kirchhoff	3..Membrane three-dimensional
Tria6 Elements	0..Default	0..Default
	3..Quadratic	3..Membrane three-dimensional
Quad4 Elements	0..Default	0..Default
	1..Linear	1..Doubly-curved thick shell
	2..Linear Reduced Hourglass control	2..Doubly-curved thin shell
	5..Linear Discrete Kirchhoff	3..Membrane three-dimensional
Quad8 Elements	0..Default	0..Default
	3..Quadratic	1..Doubly-curved thick shell
	4..Quadratic Reduced	3..Membrane three-dimensional

In addition, an existing material may then be selected from the **Membrane** drop-down and/or the **Bending** drop-down in the **Material Overrides** section.

Property Extensions - Plane Strain

MSC Nastran tab - These options are only used for SOL 400 and written to a PSHLN2 entry. When **Large Strain** is enabled, two options, **Integration Type** and **Behavior Type**, are available for each element topology.

Available options for **Integration Type** and **Behavior Type** for each Element Topology for Plane Strain:

Element Topology	Integration Type	Behavior Type
Tria3 Elements	0..Default	0..Default
	1..Linear	1..Axisymmetric Solid
		2..Incompressible Axisymmetric
		4..Plane Strain
		5..Incompressible Plane Strain
Tria6 Elements	0..Default	0..Default
	3..Quadratic	1..Axisymmetric Solid
		3..Plane Stress
		4..Plane Strain
Quad4 Elements	0..Default	0..Default
	1..Linear	1..Axisymmetric Solid
		3..Plane Stress

Element Topology	Integration Type	Behavior Type
	2..Linear Reduced Hourglass control 6..Linear with Twist	4..Plane Strain
Quad8 Elements	0..Default 3..Quadratic 4..Quadratic Reduced 5..Quadratic with Twist	0..Default 1..Axisymmetric Solid 3..Plane Stress 4..Plane Strain

In addition, an existing material may then be selected from the **Material** drop-down as well as a **Layer Direction** and/or **Thickness** in the **Overrides** section.

Property Extensions - Laminate

MSC Nastran tab - These options are only used for SOL 400 and are written to PHSLN1 entry. When **Large Strain** is activated, two options, **Integration Type** and **Behavior Type**, are available for each element topology.

Available options for **Integration Type** and **Behavior Type** for each Element Topology.

Element Topology	Integration Type	Behavior Type
Tria3 Elements	0..Default 1..Linear 5..Linear Discrete Kirchhoff	0..Default 2..Doubly-curved thin shell 3..Membrane three-dimensional
Tria6 Elements	0..Default 3..Quadratic	0..Default 3..Membrane three-dimensional
Quad4 Elements	0..Default 1..Linear 2..Linear Reduced Hourglass control 5..Linear Discrete Kirchhoff	0..Default 1..Doubly-curved thick shell 2..Doubly-curved thin shell 3..Membrane three-dimensional
Quad8 Elements	0..Default 3..Quadratic 4..Quadratic Reduced	0..Default 1..Doubly-curved thick shell 3..Membrane three-dimensional

In addition, an existing material may then be selected from the **Membrane** drop-down and/or the **Bending** drop-down in the **Material Overrides** section.

Property Extensions - Axisymmetric

MSC Nastran tab - These options are only used for SOL 400 and written to a PSHLN2 entry. When **Large Strain** is activated, two options, **Integration Type** and **Behavior Type**, are available for each element topology.

Available options for **Integration Type** and **Behavior Type** for each Element Topology for Plane Strain:.

Element Topology	Integration Type	Behavior Type
Tria3 Elements	0..Default 1..Linear	0..Default 1..Axisymmetric Solid 2..Incompressible Axisymmetric 4..Plane Strain 5..Incompressible Plane Strain
Tria6 Elements	0..Default 3..Quadratic	0..Default 1..Axisymmetric Solid 3..Plane Stress 4..Plane Strain
Quad4 Elements	0..Default 1..Linear 2..Linear Reduced Hourglass control 6..Linear with Twist	0..Default 1..Axisymmetric Solid 3..Plane Stress 4..Plane Strain
Quad8 Elements	0..Default 3..Quadratic 4..Quadratic Reduced 5..Quadratic with Twist	0..Default 1..Axisymmetric Solid 3..Plane Stress 4..Plane Strain

In addition, an existing material may then be selected from the Material drop-down as well as a Layer Direction and/or Thickness in the Overrides section.

Property Extensions - Solid

MSC Nastran tab - These options are only used for SOL 400 and written to the PSLDN1 entry.

Available options for **Integration Type** and **Behavior Type** for each Element Topology for Plane Strain:.

Element Topology	Integration Type	Behavior Type
Tetra4 Elements	0..Default	0..Default

Element Topology	Integration Type	Behavior Type
	1..Linear	1..Solid
		3..Incompressible Solid
Tetra10 Elements	0..Default	0..Default
	2..Linear Reduced Hourglass control	1..Solid
	6..Quadratic	
Pyram5 Elements	0..Default	0..Default
	1..Linear	1..Solid
	2..Linear Reduced Hourglass control	
Pyram13 Elements	0..Default	0..Default
	6..Quadratic	1..Solid
Wedge6 Elements	0..Default	0..Default
	1..Linear	1..Solid
Wedge15 Elements	0..Default	0..Default
	6..Quadratic	1..Solid
Brick8 Elements	0..Default	0..Default
	1..Linear	1..Solid
	2..Linear Reduced Hourglass control	2..Solid Shell
	3..Assumed Strain Enhanced Formulation Solid Shell	
Brick20 Elements	0..Default	0..Default
	4..Quadratic Reduced	1..Solid
	6..Quadratic	

In addition, an existing material may then be selected from the **Material** drop-down in the **Overrides** section.

Property Extensions - Solid Laminate

MSC Nastran tab - The **Brick8 Elements** and **Brick20 Elements** sections contain an **Integration Type** drop-down menus. Available options for **Brick8 Elements** are **0..Default**, **1..Linear**, and **2..Assumed Strain Enhanced For-mulation Solid Shell**, while available options for **Brick20 Elements** are **0..Default** and **3..Quadratic**.

In addition, it is possible to specify a **Layer Direction** in the **Overrides** section. The value for **Layer Direction** must be +/-1, +/-2, or +/-3 only. Default in +1. A positive value implies that the composite layer input is a fractional percent of the total element thickness in the ply direction and is recommended. A negative value implies that the composite layer input is the actual thickness of that ply.

These options, along with the options in the Define Property dialog box are used to write the PCOMPLS entry.

Property Extensions - Solid Cohesive

MSC Nastran tab - These options are only used for SOL 400 and write. The **Cohesive Elements** section contains three drop-downs.

Loads and Constraints

- Added an **Equation Coord Sys** drop-down to the Create Loads dialog box for the **Model → Load → Nodal**; **Model → Load → Nodal on Face**; **Model → Load → Elemental**; and **Model → Load → Elemental on Face** commands. The **Equation Coord Sys** drop-down can be used to specify the coordinate system in which equations are evaluated when defining nodal or elemental loads when **Method** is set to **Variable**. By default, this option is set to the active coordinate system in the model, not to match the **Definition Coord Sys**.

Connections (Regions, Properties, and Connectors)

- Added a **Check for Connection to Internal Edges** option to the **Connect → Automatic** command, which is only available when **Edge - Edge** is activated in the **Connection Types** section of the Auto Detection Options for Connections dialog box. When activated, it allows connections to be automatically created between any combination of free edges an internal edges (i.e., non-free edges) of surfaces (sheet solids), solids, and general bodies which are considered coincident based on the value specified for **Tolerance** in the **Detection Options** section. If deactivated, which is the default, no connections will be made between any edge and any internal edge.

- Added a **Surfaces to the Output** section for Connection Regions. Only available when **Defined By** is set to **Surfaces** and is currently only used by MSC Nastran.

- Updated the Connection Region Options dialog box to have three tabs: **Simcenter Nastran**, **MSC Nastran**, and **ABAQUS**. The options on the **Simcenter Nastran** and **ABAQUS** tabs were available in previous versions of Femap, but all of the options on the **MSC Nastran** tab other than **Friction Value (FRIC)** are new for 2021.1.

Body Surface Contact Flag (COPTB)

Flag to indicate how body surfaces may contact. Writes COPTB and corresponding value to BCBDPRP entry. Please see MSC documentation for BCONPRG entry for more information.

Friction Value (FRIC) or Friction Table (FRIC - TABLE)

When **Friction Value (FRIC)** is selected, specifies a friction coefficient and writes FRIC and a real number to BCBDPRP entry. When set to **Friction Table (FRIC - TABLE)**, represents ID of an existing function, which writes FRIC and an integer value to BCBDPRP.

In general, entering the friction coefficient for a contact body pair via a BCTABLE and BCONECT/ BCONPRP is strongly recommended. When a grid point contacts a rigid body, coefficient of friction

associated with the rigid body is used. When a grid point contacts a deformable body, average of the coefficients for the two bodies is used.

Smoothing Control (IDSPL)

Controls geometric smoothing of boundary of deformable body option. Enter a value of 0 or leave blank for discrete geometric representation. When a value above 0 is used, the surface of the body is smoothed out with splines (2D) or Coons surfaces (3D) and discontinuity edges/corners are being defined by using absolute values of this setting as the ID of the BLSEG entries. If a BLSEG with ID that does not equal the absolute value of this setting does not exist, the whole body is smoothed and there are no user-defined discontinuity corners (2D) or edges (3D). Furthermore, additional discontinuity edges are being generated automatically if the difference in patch normals exceeds the value of **Discontinuity Threshold (SANGLE)**. Default = 0. Writes IDSPL and corresponding value to BCBDPRP entry.

Midside Node Projection (MIDNOD)

Mid-side node projection flag. Only used when **Smoothing Control IDSPL** does not equal 0. When set to a value above 0, the midside grid of quadratic elements are projected onto the selected spline surfaces. This operation is performed before the contact process starts and it may change the location of grids in contact bodies. It may operate in combination with the initial stress-free contact. Writes MIDNOD and corresponding value to BCBDPRP entry.

Discontinuity Threshold (SANGLE)

Threshold for automatic discontinuity detection in degrees. Used for geometric smoothing option in SOL 400 only. Default = 60.0. Not used when **Smoothing Control (IDSPL)** is greater than or equal to 0. Writes SANGLE and corresponding value to BCBDPRP entry.

Contact Check (ISTYP)

Check of contact conditions and is not necessary in segment-to-segment contact. Writes ISTYP and either 0 (default) or 2 to BCBDPRP entry.

0..Check Each Body

Each deformable body is checked versus the other.

1..Optimized Contact

Double-sided contact with automatic optimization of contact constraint equations.

Rigid Options button

Opens the Rigid Region Options dialog box, which is used to specify values for the various fields of the BCRIGID entry.

Motion Control

This section is used to specify the CGID and CONTROL fields on the first line of the BCRIGID entry, along with the NLOAD field on the second line. **Control Node (CGID)** and **Rotation Control Node (NLOAD)** are the IDs of existing node(s). Choose from one of the three options for **Control Type** (CONTROL).

Rotation Axis

This section is used to specify values for the DCOS1, DCOS2, and DCOS3 fields on the second line of the BCRIGID entry. When set to **X (DCOS1)/Y(DCOS2)/Z(DCOS3)**, writes a specified value

as a real number to DCOS1, DCOS2 and/or DCOS3 fields, respectively. If set to **Time Dependent X/Time Dependant Y/Time Dependent Z**, writes ID of existing function to DCOS1, DCOS2, and/or DCOS3 fields as an integer.

Growth Factor

This section is used to specify values for the fields on the GROW line of the BCRIGID entry. The values for Component 1 (GF1), Component 2 (GF2), and Component 3 (GF3) are written to the GF1, GF2, and GF3 fields, respectively. The IDs of existing functions selected for Table 1 (TAB-GF1), Table 2 (TAB-GF2), and Table 3 (TAB-GF3) are written to the TAB-GF1, TAB-GF2, and/or TAB-GF3 fields, respectively.

Angular Velocity

This section is used to specify values for the ANGVEL on the first line of the BCRIGID entry, along with the values for the VELRB1, VELRB2, and VELRB3 fields on the second line of the BCRIGID entry. When set to **Angular Velocity Value (ANGVEL)/VX (VELRB1)/VY (VELRB2)/VZ (VELRB3)**, writes a specified value as a real number to ANGVEL, VX (VELRB1), VY (VELRB2) and/or VZ (VELRB3) fields, respectively. If set to **Time Dependent Value/Time Dependant VX/Time Dependent VY/Time Dependent VZ**, writes ID of existing function to ANGVEL, VX (VELRB1), VY (VELRB2), and/or VZ (VELRB3) fields as an integer.

Approach Velocity

This section is used to specify values for the fields on the APPROV line of the BCRIGID entry. Angular Velocity (A) writes a value to the A field. Rotation Axis X (N1)/Rotation Axis Y (N2)/Rotation Axis Z (N3) are written to the N1, N2, and N3 fields, respectively, while Velocity X (V1)/Velocity Y (V2)/Velocity Z (V3) are written to the V1, V2, and V3 fields.

- Added **Norm Scale Fac (SFNPNLT)** and **Tang Scale Fac (SFTPNLT)** options to the **Contact** section of the MSC Nastran Advanced Contact Options dialog box accessed by the **Advanced Options** button on the **MSC Nastran** tab.

Norm Scale Fac (SFNPNLT)

Scale factor of augmented Lagrange penalty factor along contact normal direction. Value must be above 0.0 (Default=1.0). `Norm Scale Fac (SFNPNLT)*Lagrange Factor (PENALT)` used in analysis. If this field is left default, value of SFNPNLT entered for BCPARA, specified in **Analysis Set Manager**, will be used. A non-blank value entered in SFNPNLT will override the value of SFNPNLT in BCPARA. SOL 400 only.

Tang Scale Fac (SFTPNLT)

Scale factor of augmented Lagrange penalty factor along contact normal direction. Value must be above 0.0 (Default=1.0). `Tang Scale Fac (SFTPNLT)*Lagrange Factor (PENALT)` used in analysis. If this field is left default, value of SFTPNLT entered for BCPARA, specified in **Analysis Set Manager**, will be used. A non-blank value entered in SFTPNLT will override the value of SFTPNLT in BCPARA. Sol 400 only.

- Added a **Cohesive Options** button to the **Glue Break** section of the MSC Nastran Advanced Contact Options dialog box accessed by the **Advanced Options** button on the **MSC Nastran** tab. Displays the Contact Glued Contact Options dialog box, which can be used to specify parameters for **Cohesive (Flexible) Glued Contact**. These options are written as parameters to a BCONPRG entry,

which is potentially used in conjunction with BCONPRP entry, with one or both entries referenced by a BCONECT entry. Currently, these options are only for SOL 400.

> Note:
>
> **Cohesive (Flexible)** contact is only supported for surface-to-surface contact, thus all limitations for sur-face-to-surface contact are also applicable to cohesive contact. In addition, the options in the **Glue Break** section are not supported when using Cohesive contact.

Enable Cohesive Contact (FGCFLG)

Flag to activate Cohesive Glued Contact. Creates PGCFLG entry. When deactivated, which is default, writes 0, which indicates no Cohesive Contact. When activated, writes 1, which activates Cohesive Glued Contact.

Normal Stiffness (FGCNST)

Equivalent normal contact stiffness of connector for Cohesive Glued Contact. Creates FGCNST entry and corresponding value.

Tangent Stiffness (FGCTST)

Equivalent tangential contact stiffness of connector for Cohesive Glued Contact. Creates FGCTST entry and corresponding value.

Normal Stress Function (FGCNSTR)

ID of function defining contact normal stress versus relative displacement for Cohesive Glued Contact. Creates FGCNSTR entry referencing ID of existing function.

Tangent Stress Function (FGCTSTR)

ID of function defining contact tangential stress versus relative displacement for Cohesive Glued Contact. Creates FGCTSTR entry referencing ID of existing function.

Normal Stiffness Function (FGCNSTI)

ID of function defining contact normal stiffness versus relative displacement or temperature for Cohesive Glued Contact. Creates FGCNSTI entry referencing ID of existing function.

Tangent Stiffness Function (FGCTSTI)

ID of function defining contact tangential stiffness versus relative displacement or temperature for Cohesive Glued Contact. Creates FGCTSTI entry referencing ID of existing function.

Reference Node (FGCRCEN)

ID of node on which the resultant contact and moment are based.

- Added an **Interference Options** button to MSC Nastran Advanced Contact Options dialog box accessed by the **Advanced Options** button on the **MSC Nastran** tab.
The options in this section are only available when **Type** is set to **0..Contact** and written as parameters to a BCONPRG entry, which can potentially be used in conjunction with a BCONPRP entry, with one or both entries referenced by a BCONECT entry. Currently, these options are only for SOL 400.

Method (OPINTRF)

The method for interference fit. Integer value greater than or equal to 0 (Default=0).

0..Small Interference Fit

Small interference fit (solved in one increment).

1..Varying with time and resolved in normal direction

Interference fit varies with time; resolved in normal direction.

2..Varying with time in a user specified direction

Interference fit varies with time in a user specified direction.

3..Varying with time using a scale factor

Interference fit varies with time using scale factor.

4..Automatic fit varying with time and node location

Automatic interference fit varies with time and node location.

Variation Table (TBINTRF)

ID of function used for interference fit. Creates TBINTRF entry referencing ID of existing function. Usage of the function depends on the setting for Method (OPINTRF):

0..Small Interference Fit

Not used.

1..Varying with time and resolved in normal direction

Variation of interference closure with respect to time.

2..Varying with time in a user specified direction

Variation of magnitude of translation vector with respect to time.

3..Varying with time using a scale factor

Variation of magnitude of scale factor with respect to time.

4..Automatic fit varying with time and node location

Variation of magnitude of penetration vector projected node position on the master segment with respect to time.

Region (CBINTRF)

Specifies if the Slave region or Master region of each connector referencing a particular connection property should be used for interference fit. Creates CBINTRF entry and writes 0 ("0..Slave" selected, default) or 1 ("1..Master" selected). Usage depends on the setting for Method (OPINTRF):

0..Small Interference Fit and 1..Varying with time and resolved in normal direction

Not used.

2..Varying with time in a user specified direction

Use translation vector of each Slave or Master region.

3..Varying with time using a scale factor

Use scale factors of each Slave or Master region.

4..Automatic fit varying with time and node location

Determines penetrations of each Slave or Master region.

Fit CSys (CDINTRF)

ID of coordinate system used by Interference Vec X (VXINTRF), Interference Vec Y (VYINTRF), and Interference Vec Z (VZINTRF) for interference fit. Creates CDINTRF entry referencing ID of existing coordinate system. Usage depends on the setting for Method (OPINTRF):

0..Small Interference Fit, 1..Varying with time and resolved in normal direction, and 4..Automatic fit varying with time and node location

Not used.

2..Varying with time in a user specified direction

Coordinate system for direction cosines of translation vector.

3..Varying with time using a scale factor

Coordinate system for scale factor vector.

Pen Search Tolerance (PTINTRF)

Penetration search tolerance for interference fit. Only used when **Method (OPINTRF)** is set to **4..Automatic fit varying with time and node location**. The value must be greater than or equal to 0 (Default = error tolerance*100).

Interference Vec X (VXINTRF), Interference Vec Y (VYINTRF), and Interference Vec Z (VZINTRF)

X, Y, and Z components of vector for interference fit (Default = 0.0). Usage depends on the setting for **Method (OPINTRF)**:

0..Small Interference Fit, 1..Varying with time and resolved in normal direction, and 4..Automatic fit varying with time and node location

Not used.

2..Varying with time in a user specified direction

Direction cosines for the translation vector.

3..Varying with time using a scale factor

Scale factor vector.

Scaling Center X (XCINTRF), Scaling Center Y (YCINTRF), and Scaling Center Z (ZCINTRF)

X, Y, and Z components interference fit center of scaling. Only used when **Method (OPINTRF)** is set to **3..Varying with time using a scale factor**. Coordinates must be specified in global rectangular coordinate system (Default = 0.0).

Initial Configuration (OPINGP)

Initial gap or overlap option. Choose from **0..None** or **1..Initial Gap or Overlap**. Writes OPINGP with corresponding value of 0 (**0..None**) or 1 (**1..Initial Gap or Overlap**).

Search Tolerance (TOLINGP)

Search tolerance of initial gap. Only used when **Initial Configuration (OPINGP)** is set to **1..Initial Gap or Overlap**. Writes TOLINGP and corresponding value (Default = average edge length of all contact segments).

Adjusted Region (CDINGP)

Specifies if Slave region or Master region of each connector referencing a particu-lar connection property should be adjusted. Only used when Initial **Configuration (OPINGP)** is set to **1..Initial Gap or Overlap**. Creates CDINGP entry, writes 0 (**0..Slave** selected, default) or 1 (**1..Master** selected).

Overlap Magnitude (MGINP)

Gap or overlap magnitude. Only used when Initial **Configuration (OPINGP)** is set to **1..Initial Gap or Overlap**. When value is greater than 0.0, signifies a gap. When value less than 0.0, signifies an overlap. When value = 0.0, which is the default, initial gap distance is preserved.

Listing

Added an **Evaluate Group to insure Entity Selection is valid** option to the **List → Groups** command. This option is on by default, but only used when the **List Entities in Group** option is enabled. Simply performs the **Group → Operations → Evaluate** command on all selected groups before listing the entities in each group.

Views

- Added support for RBE1 elements when using Best Available or Performance Graphics.

Output and Post-Processing

- Added a **Linear and Parabolic Types** option to the Contour Options dialog box, which controls if averaging of results occurs across boundaries between linear and parabolic solid elements.

Geometry Interfaces

The following Femap interfaces have been updated to support newer geometry formats:

Femap Interface	Latest Supported Version
Parasolid	33.0
NX	version 1926 (2020)
Solid Edge	2021

For details, see *Section 9, "Geometry Interfaces"* in the Femap User Guide.

Analysis Program Interfaces

Several of the analysis program interfaces have been improved. These changes include:

- Femap Neutral File Interface

- MSC Nastran Interface

- Simcenter Nastran Interface

- Nastran Interfaces (Simcenter and MSC/MD)

- ABAQUS Interface

For details, see *Section 8, "Analysis Program Interfaces"* in the Femap User Guide.

Femap Neutral File Interface

- Updated Neutral Read and Write for v2021.1 changes.

MSC Nastran Interface

SOL 400 Overview

- Added support for Analysis Type **30..Implicit Nonlinear**, which is used to setup analysis jobs using SOL 400. To support SOL 400, new dialog boxes have been created in the **Analysis Set Manager** to enter options for Non-linear Behavior (Nonlinear Behavior Options), Automatic Contact Parameters (Automatic Contact Generation), and Control Parameters (NASTRAN Implicit Nonlinear Analysis). In addition, new options were added to Contact Parameters (MSC Nastran Contact Solver Parameters). Finally, two new entity types, Analysis Step (Analysis Step dialog box) and Global Step (Global Analysis Steps dialog box), have been created to perform analysis chaining within a particular subcase.

Nonlinear Behavior

The Nonlinear Behavior Options dialog box is used to specify values for specific fields on the NLMOPTS entry:

When on, **Enable NLMOPTS** writes specified options to NLMOPTS entry. When off, writes nothing.

Nonlinear Properties

The items in this section are written to the SPROPMAP line of the NLMOPTS entry. These options control auto-mated inclusion of nonlinear property options. See MSC Nastran Quick Reference Guide for more information.

Mapping (PROPMAP)

Value written to PROPMAP field depends on selected option.

-1..No mapping considered

Writes a value of −1, which means no mapping is considered.

0..Full integration elements when necessary

Writes 0, which means map to full integration elements associated with nonlinear property extension entries when necessary.

1..Reduced integration elements when necessary

Writes 1, which means map to reduced integration elements associated with nonlinear property extension entries when necessary.

2..Full integration elements when possible

Writes 2, which means map to full integration elements associated with nonlinear property extension entries when possible.

3..Reduced integration elements when possible

Writes 3, which means map to reduced integration elements associated with nonlinear property extension entries when possible.

Behavior (PROPBEH)

Text written to PROPBEH field depends on selected option. **0..1D** writes 1D, **1..3D** writes 3D, **2..PLST** writes PLST, **3..PSTR"** writes PSTR, and **4..AXI** writes AXI.

Print Flag (IPRINT)

Print flag for generated property. Value written to IPRINT field depends on selected option.

0..Default

Writes value of 0.

1..Print

Writes 1.

2.Punch

Writes 2.

3..Print and Punch

Writes 3.

Layer Direction (DIRECT)

Layer Direction Flag. Value written to DIRECT field depends on specified value. Must be an integer value of 1, 2, or 3. Default is 1.

Thickness (THICKOP)

Out of plane thickness for 2D solids. Value written to THICKOP field depends on specified value, which must be greater than or equal to 0.0. Default is 1.0.

Composite Options

The first two items in this section are written to the INLAM line of the NLMOPTS entry, while the third is written to the SHEAR line.

Output (VCOORD)

Selects coordinate system for interlaminar stress output. Text written to VCOORD field depends on selected option.

0..Use original CSys

Writes nothing, meaning use original coordinate system for interlaminar stress output.

1..Use Ply CSys of bottom layer

Writes BOTT, meaning use ply coordinate system of bottom layer of the interlaminate for interlaminar stress output.

2..Use Ply CSys of above layer

Writes TOP, meaning use ply coordinate system of top layer of the interlami-nate for interlaminar stress output.

X-axis Projection (CPROJ)

Activates projection system for projecting X-axis of CORDM onto layer plane. When on, writes ON to CPROJ field, which indicates projection scheme activated. When off, writes nothing or OFF.

Parabolic Shear Distribution (TSHEAR)

When on, writes TSHEAR to VALS field to indicate parabolic shear distribution is used for certain element types. When off, THEAR line is not written.

Strain Formulations

The first item in this section is written to the LRGSTRN line of the NLMOPTS entry, while the second is written to the ASSM line.

Large Strain (LRGSTRN)

pecifies Large Strain Formulation. Value written to VALLE field on LRGSTRAN line depends on selected option.

-1..No Large Strain Formulation

Writes −1.

0..Mean Normal Return

Writes 0.

1..Hypoelasticity and additive plasticity with mean normal return

Writes 1.

2..Hyperelasticity and multiplicative plasticity with radial return

Writes 2.

Assumed Strain (ASSM)

Assumed strain option. Text written to VALA field on ASSM line depends on selected option.

0..Default

ASSM line not written and assumed strain is calculated automatically for certain element types under certain conditions.

1..Use assumed strain formulation

Writes ASSUMED, which means assumed strain is calculated manually for certain element types under certain conditions.

2..Do not use assumed strain formulation

Writes OFF, which means assumed strain is never calculated.

Element Deactivation

The items in this section are written to the DEACTEL line of the NLMOPTS entry.

Rules (VALD1)

Select default deactivation rules for elements in each physics pass in the analysis. Value written to VALD1 field depends on selected option.

-1..Default deactivation rules are always ignored

Writes -1, which is default for single physics analysis.

0..Default deactivation rules are always applied

Writes 0, which is default in coupled analysis.

1..Default deactivation rules are applied in a mechanical pass only

Writes 1.

2..Default deactivation rules are applied in a thermal pass only

Writes 2.

Diag. Output (VALD2)

Select diagnostic output behavior for active and deactivated elements. Value written to VALD2 field depends on selected option.

0..Summary of deactivated elements

Writes 0, meaning a summary of all deactivated elements is printed. If no elements get deactivated output is skipped.

1..Detailed list of deactivated elements

Writes 1, meaning a detailed list of all deactivated elements is printed. If no elements get deactivated output is skipped.

2..Summary of deactivated and remaining elements

Writes 2, meaning a summary of all deactivated and all remaining active elements is printed.

3..Detailed list of deactivated and remaining elements

Writes 3, meaning a detailed list of all deactivated and all remaining active elements is printed.

Creep

The items in this section are written to the CREEP line of the NLMOPTS entry.

Enable Creep Formulations (CREEP)

When on, the CREEP line is written with either values for VALC1 and VALC2 or values for VALC3 and VALC4. This line indicates the formulation to use for creep analysis.

> Note:
>
> Creep data should be entered using the MATVP entry, not the CREEP bulk data entry.

Implicit Creep
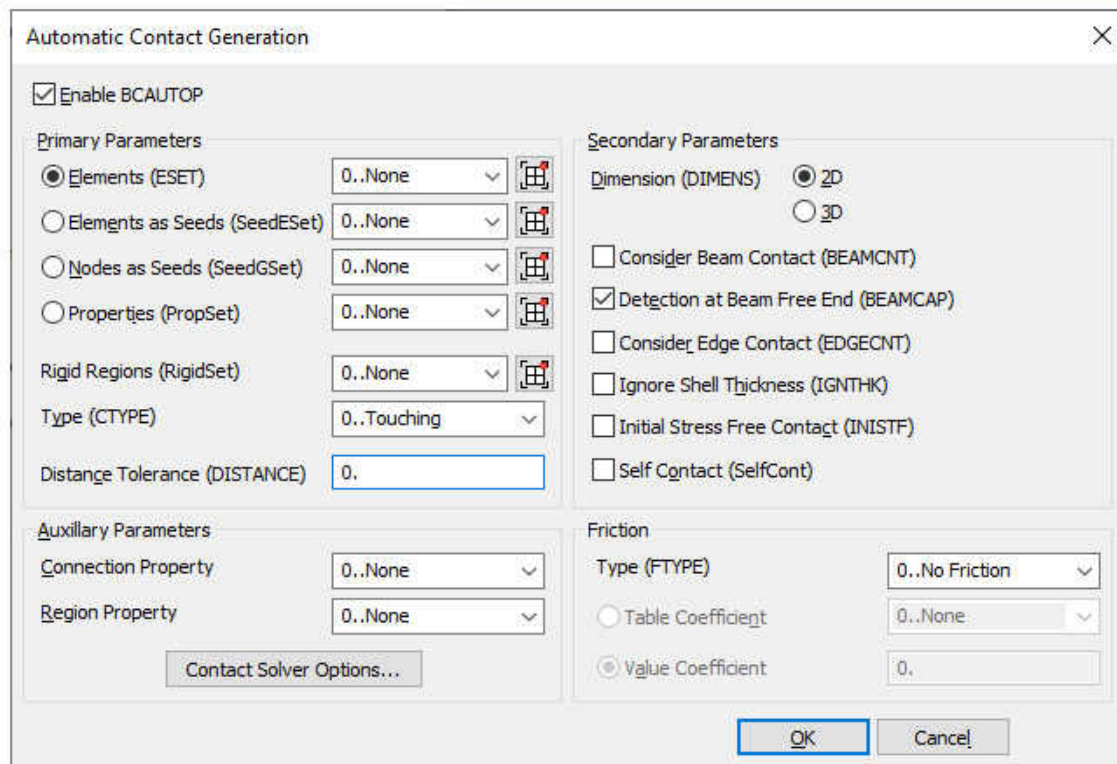
When selected, writes the selected option to a combination of the VALC3 and VALC4 fields. In all cases, a value of a value of `1` is written to VALC3. The value of VALC4 depends on selected option, where **0..Elastic Tangent** writes a value of `0`, **1..Secant Tangent** writes a `1`, or **2..Radial Return** writes a `2`.

Explicit Creep

When selected, writes the selected option to a combination of the VALC1 and VALC2 fields. In all cases, a value of a value of `0` is written to VALC2. The value of VALC1 depends on selected option, where **0..Maxwell Model Creep** writes a value of `0` or **1..Explicit Viscoplastic Creep** writes a `1`.

Automatic Contact Parameters

The Automatic Contact Generation dialog box is used to specify values for specific fields on the BCAUTOP entry, which are used for Automatic contact generation (ACG).

When enabled, **Enable BCAUTOP** writes specified options to BCAUTOP entry, When disabled, writes nothing.

Primary Parameters

The items in this section are the most important parameters for build of contact bodies, match of contact pairs.

Elements (ESET)

When selected, specifies the element IDs to only consider for automatic creation of contact bodies. When a group is selected from the drop-down, a SET1 entry will be created in Case Control containing element IDs, then ESET will be written and reference the ID of the SET1 entry. If the drop-down is set to **0..None**, then a SET1 entry will not be written or referenced by ESET, thus all elements in the model will be considered.

Elements as Seeds (SeedESet)

When selected, specifies the element IDs to consider as seeds for automatic creation of contact bodies. When a group is selected from the drop-down, a SET1 entry will be created in Case Control containing element IDs, then SeedESet will be written and reference the ID of the SET1 entry. If the drop-down is set to **0..None**, then a SET1 entry will not be written or referenced by SeedESet, thus all elements in the model will be considered as seed elements.

Nodes as Seeds (SeedGSet)

When selected, specifies node IDs to determine seed elements for automatic creation of contact bodies. Only considers elements which are connected to the specified node IDs. When a group is selected from the drop-down, a SET1 entry will be created in Case Control containing node IDs, then SeedNSet will be written and reference the ID of the SET1 entry. If the drop-down is set to **0..None**, then a SET1 entry will not be written or referenced by SeedNSet, thus all nodes in model are considered to determine seed elements.

Properties (PropSet)

When selected, specifies the property IDs to use for automatic creation of contact bodies. When a group is selected from the drop-down, a SET1 entry will be created in Case Control containing property IDs, then PropSet will be written and reference the ID of the SET1 entry. If the drop-down is set to **0..None**, then a SET1 entry will not be written or referenced by PropSet, thus all properties in model are considered.

Rigid Regions (RigidSet)

Specifies connection region IDs with **Type** set to **Rigid** to consider for automatic creation of contact bodies. When a group is selected from the drop-down, a SET1 entry will be created in Case Control containing connection regions IDs, then RigidSet will be written and reference the ID of the SET1 entry. If the drop-down is set to **0..None**, then a SET1 entry will not be written or referenced by RigidSet, thus all connec-tion regions in model with **Type** set to **Rigid** are considered.

> Note:
>
> To quickly create a new group or edit an existing group, click the **New Group** icon button next to a drop-down, then use the Quick Group dialog box to create, then populate the group with entities (**New Group** button, followed by **Edit Group** button) or simply edit an existing group (select a group from list, click **Edit Group** button). In all cases, clicking **Edit Group** will open a standard selection dialog box allowing selection of the type of entity

needed by a particular set. For instance, when using the icon but-ton next to the
Elements (ESET) drop-down, you will be prompted to select elements.

Type (CTYPE)

Type of contact. Text written to CTYPE field depends on selected option.

0..Touching

Writes TOUCH (default).

1..Permanent Glue

Writes PGLUE.

2..Step Glue

Writes SGLUE.

3..General Glue

Writes GGLUE.

Distance Tolerance (DISTANCE)

Distance tolerance of contact pair. If the distance between any two points which are belonging
to two different contact bodies is less than this value, these two contact bodies are recognized
to be a contact pair. Default value of DISTANCE is 100 times value of contact tolerance (ERROR).
If using the default ERROR, the value for ERROR is the smallest of the following values:

- 1/20 of the smallest nonzero element dimension (plates or solids) in the contact body.

- 1/4 of the thinnest shell thickness in the contact body.

Auxiliary Parameters

The items in this section allow for selection of an existing connection property and/or connection
region from the appropriate drop-down to specify options on the BCAUTOP entry.

In addition, the **Contact Solver Options** button displays the MSC Nastran Contact Solver Parameters
dialog box. For more information, see *Section 8.8.1.33, "Contact Parameters (MSC Nastran Only)"* in
the Femap User Guide.

Secondary Parameters

The items in this section are helpful for user to provide more detail and further requirement for
contact model establishment and contact analysis with automatic contact generation.

Dimension (DIMENS)

Dimension of contact bodies to create. When set to 2D, writes DIMENS and 2D. When set to 3D
(default), writes DIMENS and 3D.

Consider Beam Contact (BEAMCNT)

Determines if beams should be included in contact definition. When enabled, writes BEAMCNT and YES. When disabled, which is the default, writes nothing.

Detection at Beam Free End (BEAMCAP)

Mainly used for contact pair detection where contact occurs at free beam end. When enabled, which is the default, writes nothing. When disabled, writes BEAMCAP and NO.

Consider Edge Contact (EDGECNT)

Determines if shell edge is included in contact definition. When enabled, writes EDGECNT and YES, which means free and hard shell edges are included in the contact definition. When disabled, which is the default, writes nothing and no shell edges are included in the contact definition.

Ignore Shell Thickness (IGTHNK)

Indicates if shell element thickness should be ignored for contact detection. When enabled, writes IGNTHK and YES to ignore shell thickness. When disabled, which is the default, writes nothing and shell thickness is included in detection.

Initial Stress Free Contact (INISTF)

Indicates if initial stress free contact should is allowed. When enabled, writes INISTF and YES, meaning initial stress free contact is implemented. When disabled, which is the default, writes nothing, meaning general contact without implementation of initial stress free.

Self Contact (SelfCont)

Indicates if self contact should be considered. When enabled, writes SelfCont and Yes. When disabled, writes nothing.

Friction

The items in this section are used to specify the options for friction.

Type (FTYPE)

Type of friction. When set to **0..No Friction**, which is the default, writes nothing. When set to **6..Bilinear Coulomb Friction**, writes FTYPE and BLCOUL (indicates value of 6). When set to **7..Bilinear Shear Friction**, writes FTYPE and BLSHEAR (indicates value of 7).

Coefficient options

Used to specify value for friction. When set to **Table Coefficient**, writes FRIC along with an integer value which corresponds to the ID of function currently selected in the drop-down. When set to **Value Coefficient**, writes FRIC along with the value specified in the field.

NASTRAN Implicit Nonlinear Analysis

Opens the Nastran Implicit Nonlinear Analysis dialog box, which is used to specify control parameters for the Master Case, any subcase, or any analysis step for implicit nonlinear analysis (SOL 400):

Writes specified options to NLSTEP Bulk Data entry, along with NLSTEP = ID entry to the appropriate location in Case Control. If **Skip NLSTEP** is on, no options may be specified and nothing is written.

Value for ID is written to the ID field of NLSTEP. If a Subcase or Step has the same ID as Master Case, a different Subcase, and/or a different Step, it will be properly incremented during export, so each NLSTEP has a unique ID.

Step Parameters	The items in the section are written to specific fields on the first line of the NLSTEP entry.

Total Time (TOTTIM)

Total time for the load case. Must be a real number with a positive value. Default is 1.0. Value is written to TOTTIM field.

Nonlinearity (CTRLDEF)

Used to control the default values used for options specified the **General Parameters** section and the Mechanical Analysis Parameters dialog box, which is accessed by clicking the **Mechanical Parameters** button.

> Note:
>
> Once an option has been selected, click the **Defaults** button in the appropriate dialog box to update the default values. This will allow blank values to be written to the input file for those options.

Writes selected option to CTRLDEF field.

0..Default

Writes nothing.

1..Linear

Writes QLINEAR, meaning defaults are set for quasi-linear analysis.

2..Mildly Nonlinear

Writes MILDLY, meaning defaults are set for mildly nonlinear analysis.

3..Severely Nonlinear

Writes SEVERELY, meaning defaults are set for severely nonlinear analysis.

General Parameters

The items in the section are written to specific fields on the GENERAL of the NLSTEP entry.

Max Iterations / Increment (MAXITER)

Maximum number of iterations allowed for each increment. Default is 10. Writes value to MAXITER field.

Min Iterations / Increment (MINITER)

Minimum number of iterations needed for each increment. Must be a value greater than 0. Default is 1, unless **Nonlinearity** (CTRLDEF) is set to **2..Severely Nonlinear**, then default value is 2. Writes value to MINITER field.

Max Bisections for Step (MAXBIS)

Maximum number of bisections allowed in the current step. Value can be positive or negative. Default is 10. If value is positive and solution does not converge after the specified number of bisections, the best solution is computed and the analysis is continued to the next time step. If value is negative and solution does not converge after the absolute value of specified number of bisections, analysis is terminated.

Include Creep (CREEP)

Flag to include creep in current load case. If disabled, which is the default, nothing is written to the CREEP field, meaning creep will not be included, even if creep is defined for a material. When on, a value of 1 is written to the CREEP field. For nonlinear elements only..

Analysis Parameters

Depending on the setting for **Increment Type**, the different options are available when clicking the **Increment Parameters** button.

When **Increment Type** is set to **0..Fixed**, clicking the **Increment Parameters** button displays the Fixed Time Stepping Parameters dialog box. See *Fixed Time Stepping Parameters* in the Femap User Guide for more information.

When **Increment Type** is set to **1..Adaptive**, clicking the **Increment Parameters** button displays the Adaptive Load Stepping Parameters dialog box. See *Adaptive Load Stepping Parameters* in the Femap User Guide for more information.

When **Increment Type** is set to **2..Arc-Length**, clicking the **Increment Parameters** button displays the Arc-Length Load Stepping Parameters dialog box. See *Arc-Length Load Stepping Parameters* in the Femap User Guide for more information.

> Note:
>
> The arc-length increment type is not supported for contact analysis and should not be used when **Analysis Type** if set to **3..Transient**.

Clicking the **Mechanical Parameters** button displays the Mechanical Analysis Parameters dialog box. See *Mechanical Analysis Parameters* in the Femap User Guide for more information.

Fixed Time Stepping Parameters

The items in Fixed Time Stepping Parameters dialog box are written to the FIXED line of the NLSTEP entry.
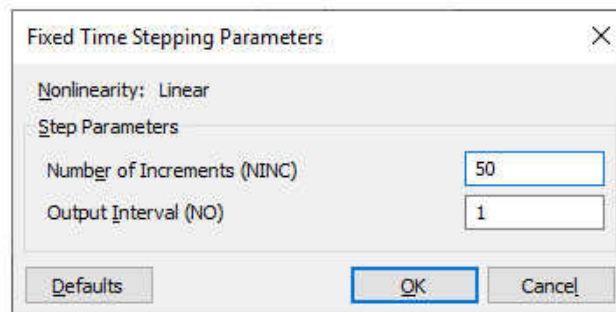
Step Parameters

Number of Increments (NINC)

Number of increments for fixed time stepping. Value must be above 0. Default is 50. When value other than 50 is entered, writes value to NINC field. The time step for each increment is

calcu-lated value specified for Total Time (TOTTIM)/value specified for Number of Increments (NINC).

Output Interval (NO)

Interval for output. Every NO-th increment will be saved for output. Must be a value greater than or equal to 0. Default is 1. When value other than 1 is entered, writes value to NO field.

Adaptive Load Stepping Parameters

The items in Adaptive Load Stepping Parameters dialog box are written to the various ADAPT lines of the NLSTEP entry.

Time Steps

Initial Fraction (DTINITF)

Initial time step defined as fraction of Total Time (TOTTIM). Default is 0.01. If this value is greater than or equal to **Max Fraction (DTMAXF),** then the value is reset to that value. If **Nonlinearity (CTRLDEF)** is set to **1..Linear**, this value should be set equal to **Total Time**

(TOTTIM). When set to a value other than 0.01, writes specified value to DTINITF filed on first ADAPT line.

Min Fraction (DTMINF)

Minimum time step defined as fraction of **Total Time (TOTTIM)**. Default is 1.0E-5. When set to a value other than 1.0E-5, writes specified value to DTMINF field on first ADAPT line.

Max Fraction (DTMAXF)

Maximum time step defined as fraction of **Total Time (TOTTIM)**. Default is 0.5. When set to a value other than 0.5, writes specified value to DTMAXF field on first ADAPT line.

> Note:
>
> When **Analysis Type** is set to **3..Transient** for the Master Case, Subcase, or Step, this value should be set between 0.05 and 0.2.

Increase Factor (SFACT)

Factor for increasing time steps due to number of iterations. Default is 1.2. When set to a value other than 1.2, writes specified value to SFACT field on first ADAPT line.

Auto Skip Factor (ADJUST)

Time step skip factor for automatic time step adjustment. Only used when **Analysis Type** is set to **3..Transient** for the Master Case, Subcase, or Step. Default is 0. When set to a value other than 0, writes value to ADJUST field on third ADAPT line.

Dominant Period Resp. Steps (MSTEP)

Number of steps to obtain the dominant period response. Default is 10. When value other than 10 is specified, writes value to MSTEP field on third ADAPT line.

When set to an integer value from 10 to 200, activates frequency based adaptive time step adjustment with bounds, using value specified for Adaptive Time Step Bound (RB), when specified increment converges. It cannot be combined with iteration based adaptive time step adjustment, controlled by Desired Iterations / Increment (NDESIR), thus the value for Desired Iterations / Increment (NDESIR) is always ignored in this case.

When set to a value of -1, activates frequency based adaptive time step adjustment without bounds, thus value specified for Adaptive Time Step Bound (RB) is ignored. It can be combined with iteration based adaptive time step adjustment controlled by Desired Iterations / Increment (NDESIR). If Desired Iterations / Increment (NDESIR) is set to 0, time step is only adjusted after the increment converges.

Adaptive Time Step Bound (RB)

Define bounds for maintaining the same time step for the stepping function during the adaptive process. Must be value from 0.1 to 1.0. Default is 0.6. When value other than 0.6 is specified, writes value to RB field on third ADAPT line. See definition for Dominant Period Resp. Steps (MSTEP) for more information about when this value is used.

Output Control

Output Flag (INTOUT) and Output Every Nth Step

Output flag and Interval value to use when options is set to **1..Equally Spaced Intervals**. Writes value for selected option to INTOUT field on first ADAPT line.

-1..Last Increment

Writes value of −1, meaning only the last increment of the step will be output.

0..Every Load Increment
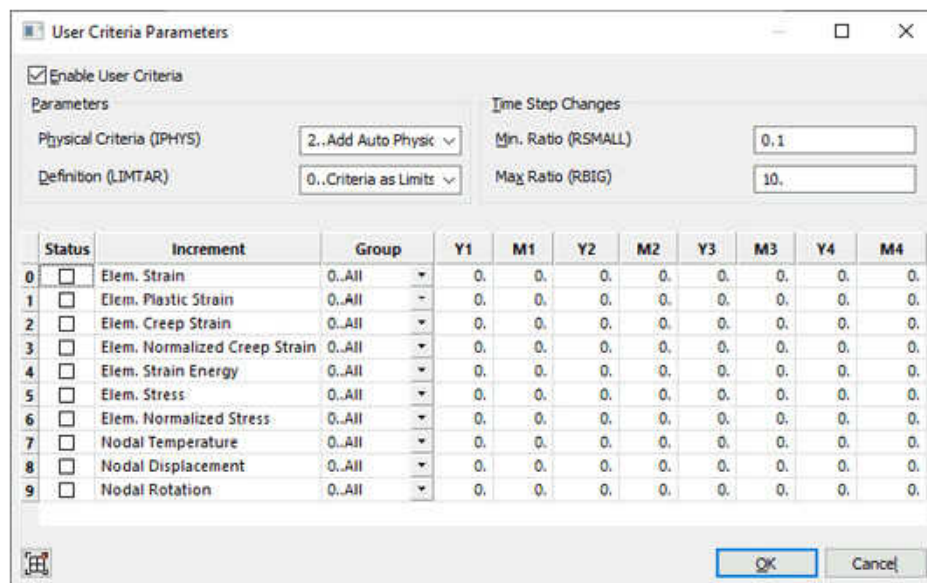
Writes value of 0, meaning every computed load increment will be output.

1..Equally Spaced Intervals

Writes, value specified for **Output Every Nth Step**, meaning output will be obtained at equally spaced intervals, thus time step will be temporarily adjusted if necessary in order to reach certain intervals.

Criteria button

Opens User Criteria Parameters dialog box:

If **Enable User Criteria** is enabled, the options in the **Parameters** and **Time Step Changes** sections in the top half of the dialog box can be used to write values to fields on the second ADAPT line of the NLSTEP entry.

Physical Criteria (IPHYS)

Determines if automatic physical criteria should be added and how analysis should proceed if a user criterion is not satisfied. Writes value for selected option to IPHYS field. Default is **2..Add Auto Physical Criteria; Stop When User Criteria Exceeded**.

-2..No Auto Physical Criteria; Continue When User Criteria Exceeded

Writes value of −2, meaning do not add automatic physical criteria; continue when user criteria are not satisfied.

-1..Add Auto Physical Criteria; Continue When User Criteria Exceeded

Writes value of −1, meaning add automatic physical criteria; continue when any user criterion is not satisfied.

1..No Auto Physical Criteria; Stop When User Criteria Exceeded

Writes value of 1, meaning add automatic physical criteria; stop when any user criterion is not satisfied.

2..Add Auto Physical Criteria; Stop When User Criteria Exceeded

Writes value of 2, meaning do not add automatic physical criteria; stop when any user criterion is not satisfied

Definition (LIMTAR)

Controls how user criteria are used. When set to **0..Criteria as Limits**, which is default, writes value of 0 to LIMTAR field. When set to **1..Criteria as Targets**, writes value of 1 to LIMTAR field.

> Note:
>
> When used as limits, time step will be reduced if the criterion would be violated. When used as targets, time step will be increased if the calculated results are less than what is specified by the criteria.

Min. Ratio (RSMALL)

Smallest ratio between time step changes due to user criteria. Default is 0.1. When set to value other than 0.1, writes value to RSMALL field.

Max Ratio (RBIG)

Largest ratio between time step changes due to user criteria. Default in 10.0. When set to value other than 10.0, writes value to RBIG field.

The bottom half of the dialog box can be used to specify values for up to 10 different user-defined criteria. These values are then written to a TABSCTL entry. The ID of the TABSCTL entry is then referenced by the CRITTID field on the second "ADAPT" line of the NLSTEP entry.

The various columns are used for several reasons, including selection of user criteria to consider, choose which entities should be considered for each user criteria, and specify limit values for each criteria.

Status

When the box in the **Status** column is on, that User Criteria will be written to the TABSCTL entry using the specified group information and ranges of limit target values.

Increment

This column contains names of the 10 potential user criteria. Selected user criteria are written as text to a ICRITi field on the TABSCTL entry. Available options are **Elem. Strain** (writes ESI),

Elem. Plastic Strain (writes EPSI), **Elem. Creep Strain** (writes ECSI), **Elem. Normalized Creep Strain** (writes ENCSI), **Elem. Strain Energy** (writes ESRI), **Elem. Stress** (writes ESTRI), **Elem. Normalized Stress** (writes ENSTRI), **Nodal Temperature** (writes GTI), **Nodal Displacement** (writes GDI), and **Nodal Rotation** (writes GRI).

Group

By default, set to **0..All**, which means all nodes in the model will be considered for the 3 nodal criteria or all elements will be considered for the 7 elemental criteria. If a group is selected, only the nodes/elements in that group will be considered by writing a SET3 bulk data entry containing node/element IDs. Each SET3 entry is then referenced by an appropriate SET3_IDi field on the TABSCTL entry.

> Note:
>
> To quickly create a new group or edit an existing group, click the **New Group** icon button in the lower left of the dialog box, then use the Quick Group dialog box to create, then populate the group with enti-ties (**New Group** button, followed by **Edit Group** button) or simply edit an existing group (select a group from list, click **Edit Group** button). In all cases, clicking **Edit Group** will open a standard selection dialog box allowing selection of the type of entity needed by a particular user criteria.

Y1, M1, Y2, M2, Y3, M3, Y4, M4

For each user criterion, up to 4 ranges of target/limit values can be specified. This allows the use of different limit/target values for different ranges of the corresponding total quality. Writes all YT1_i, MT1_i, YT2_i, MT2_i, YT3_i, MT3_i, YT4_i, and MT4_i entries needed for each selected user criteria.

Arc-Length Load Stepping Parameters

The items in the Arc-Length Load Stepping Parameters dialog box are written to the various ARCLN lines of the NLSTEP entry.

Constraint Type

Used to select the arc-length method. Writes text for selected option to TYPE field. Default is **0..Crisfield**.

0..Crisfield

Writes CRIS, which selects the Crisfield arc-length method.

1..Riks

Writes RIKS, which selects the Riks arc-length method.

2..Modified Riks

Writes MRIKS, which selects the modified Riks arc-length method.

Initial Time Step Fraction (DTINITFA)

Initial time step defined as a fraction of the **Total Time (TOTTIM)** for the arc-length procedure. Default is 0.01. If value other than 0.01 is specified, writes value to DTINITFA field.

Min ArcLen Adjust Ratio (MINALR)

Minimum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. Must be a value from 0.0 to 1.0. Default = 0.25. If value other than 0.25 is specified, writes value to MINALR field.

Max ArcLen Adjust Ratio (MAXALR)

Maximum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. Must be greater than or equal to 1.0, Default = 4.0. If value other than 4.0 is specified, writes value to MAXALR field.

Scale Factor (SCALEA)

User specified scaling factor. If any value other than 0.0 is specified, writes value to SCALEA field.

Desired Iterations (NDESIRA)

Desired number of iterations for convergence to be used for the adaptive arc-length adjustment. Must be an integer value greater than 0, Default = 4. If value other than 4 is specified, writes value to NDESIRA field.

Max Increments (NSMAXA)

Maximum number of increments in the current load case. Must be an integer value. Default is 1000. If value other than 1000 is entered, writes value to NSMAXA field.

Mechanical Analysis Parameters

The items in the Mechanical Analysis Parameters dialog box are written to the various MECH lines of the NLSTEP entry.

When **Enable MECH** is on, writes specified options to MECH lines. When disabled, writes nothing.

Based on the setting for **Nonlinearity (CTRLDEF)** in the Nastran Implicit Nonlinear Analysis dialog box, which is listed at the top, clicking the **Defaults** button populates the fields with the recommended default values.

Stiffness Updates

Method (KMETHOD)

Method for controlling stiffness updates. When set to **0..Full Newton-Raphson**, which is the default, nothing is written. When set to **1..Quasi Newton-Raphson**, writes ITER to KMETHOD field.

Iteration Interval (KSTEP)

Only available when **Method (KMETHOD)** is set to **1..Quasi Newton-Raphson**. Specifies number of iterations before the stiffness update. Default is 10. Writes value other than 10 to KSTEP field.

Advanced

ON Vectors (MAXQN)

Only used by **Method (KMETHOD)** is set to **1..Quasi Newton-Raphson**. Maximum number of quasi-Newton correction vectors to be saved on the database. Must be an integer value. Default is value for **Max Iterations / Increment (MAXITER)**. Writes value to MAXQN field.

Max Line Searches (MAXLS)

Only used by **Method (KMETHOD)** is set to **1..Quasi Newton-Raphson**. Maximum number of line searches allowed for each iteration. Default is 4. Writes value other than 4 to MAXLS field.

Line Search Tol (LSTOL)

Only used by **Method (KMETHOD)** is set to **1..Quasi Newton-Raphson**. Line Search tolerance. Must be value from 0.01 to 0.9. Default is 0.5. Writes value other than 0.5 to LSTOL field.

Fraction Limit (FSTRESS)

Fraction of effective stress used to limit the sub-increment size in material routines. Must be value from 0.0 to 1.0. Default is 0.2. Writes value other than 0.2 to FSTRESS field.

Convergence Tolerances

Displacement, Load, Work, Vector, Length, Automatic Switching

Error tolerances for displacement, load, work, vector, and length. Depending on the options enabled, writes the CONV entry with U (Displacement), P (Load), W (Work), V (Vector), L (Length), and/or A (Automatic Switching) in any combination. Default for CONV for SOL 400 for structural analysis is PV. Along with CONV, a corresponding entry and value will be written for each tolerance. When on, **Displacement** writes EPSU, **Load** writes EPSP , and **Work** writes EPSW. Default values depend on Nonlinearity (CTRLDEF) setting in the Nastran Implicit Nonlinear Analysis dialog box

Checking (MRCONV)

Specifies if rotations and moments should be included in the convergence testing. Writes value of selected option to MRCONV field. Default is **3..Forces and Displacements**.

0..Forces, Moments, Displacements, and Rotations

Writes value of 0, meaning check forces, moments, dis-placements and rotations.

1..Forces, Moments, and Displacements

Writes value of 1, meaning check forces, moments, and displacements.

2..Forces, Displacements, and Rotations

Writes value of 2, meaning check forces, moments, and rotations.

3..Forces and Displacements

Writes value of 3, meaning check forces and displacements only.

MSC Nastran Contact Solver Parameters

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.

Enable Contact Parameters must be on for Femap to use any options in this dialog box:

BCPARA

The items in this section are written to the BCPARA entry. For more information, see the BCPARA (Contact Parameters - SOLs 101 and 400) entry in the MSC Nastran Quick Reference Guide.

Contact Method (METHOD)

Contact Method. When set to **0..NODESURF**, which is the default, writes nothing, indicating node to segment contact. When set to **1..SEGTOSEG**, writes METHOD and SEGTOSEG, indicating segment-to-segment contact.

Touch Dist (ERROR)

Distance below which a node is considered touching a body. Default = 0.0 (nothing is written, meaning automatic calculation). When value above 0.0 is specified, writes ERROR and value.

Cont Tolerance (BIAS)

Contact tolerance bias factor. Value must be between 0.0 and 1.0. Default value is 0.9. When value is not 0.9, writes BIAS and value.

Max Slide Dist (SLDLMT)

Maximum allowed sliding distance, beyond it the contact segments are redefined, for segment to segment contact analysis with large deformation. Default is 0.0. If value other than 0.0 is specified, writes SLDLMT and value.

Max Slip Dist (STKSLP)

Maximum allowable slip distance for sticking, beyond it there is no sticking, only sliding exists. Only used when **Contact Method** (METHOD) is set to **1..SEGTOSEG**. Default is 0.0. If value other than 0.0 is specified, writes STKSLP and value.

All Glue (NGLUE)

Determines if the same glued behavior will be used for all subcases, which is the default, or only the first load case (first subcase and first step). If off, which is the default, and all slave regions for connectors (BCTABLE or BCONPRG entries) corresponding to the first load case (first subcase and first step) contain a value for TYPE(IGLUE) above 0 on the Connection Property, permanent glued contact with small rotation condition will be used for all slave connection regions in all subcases. If instead, then value for TYPE(IGLUE) is below 0 on the Connection Property, then permanent glued contact for large deformation and large rotation is activated. If enabled, writes NGLUE and a value of 1, meaning permanent glued contact conditions will be used for all "slave connection regions in only the first load case (first subcase and first step).

Error by Pair (ERRBAS)

Error computation option. When off, which is the default, error is computed globally for all contact pairs (connectors). When on, writes ERRBAS and value of 1, meaning the error is calculated for each contact pair (connector) individually.

Include 3D Beam (BEAMB)

Beam-Beam contact flag. When off, which is the default, nothing is written and there is no beam-to-beam contact. When on, writes BEAMB with a value of 1 and beam-to-beam contact is activated.

Ignore Tolerance Thickness (THKOFF)

Ignore thickness from the tolerance used when **Search** (ISEARCH) is set to **2..Auto** on the Connection Property when **Contact Method** (METHOD) is set to **0..NODESURF** or from the characteristic length calculated from the combination of the values for **Aug Pen Dist** (AUGDIST) and **Augment PenFactor** (PENALT) in **Contact Method** (METHOD) is set to **1..SEGTOSEG**. If off, which is the default, writes nothing, meaning do not ignore thickness. If on, writes THKOFF and value of 1, meaning remove thickness.

Augment Method (AUGMENT)

Augmentation method used in a segment-to-segment contact analysis. Writes AUGMENT and value of selected option. **0..None** is default.

0..None

Writes 0, meaning no augmentation.

1..Linear-Constant, Quadratic-Bilinear

Writes 1, meaning Augmentation based on a constant Lagrange multiplier field for linear elements and on a (bi)linear Lagrange multiplier field for quadratic elements.

2..Constant

Writes 2, meaning augmentation based on a constant Lagrange multiplier field.

3..Bilinear

Writes 3, meaning augmentation based on a (bi)linear Lagrange multiplier field.

Aug Pen Dist (AUGDIST)

Penetration distance beyond which an augmentation will be applied. Only used when **Contact Method** (METHOD) is set to **1..SEGTOSEG**. Must be a value above 0. Writes AUGDIST and value. See MSC Nastran Quick Reference Guide for more information on proper usage.

Augment PenFactor (PENALT)

Augmented Lagrange penalty factor. Only used when **Contact Method** (METHOD) is set to **1..SEGTOSEG**. Must be a value above 0.0. Writes PENALT and value. See MSC Nastran Quick Reference Guide for more information on proper usage.

Stick Pen Fact (TPENALT)

Augmented Lagrange penalty factor for sticking part of friction. Only used when **Contact Method** (METHOD) is set to **1..SEGTOSEG**. Must be a value above 0.0. The default is value specified for **Augment PenFactor (PENALT)**/1000. Writes TPENALT and value.

Normal Scale Factor (SFNPNLT)

Scale factor of augmented Lagrange penalty factor along contact normal direction. Only used when **Contact Method** (METHOD) is set to **1..SEGTOSEG**. Must be a value above 0.0 and default is 1.0. **Normal Scale Factor (SFNPNLT)** value ***Augment PenFactor (PENALT)** value is used in analysis. Writes SFNPNLT and value.

Tangent Scale Factor (SFTPNLT)

Scale factor of augmented Lagrange penalty factor along contact tangential direction. Only used when **Contact Method** (METHOD) is set to **1..SEGTOSEG**. Must be a value above 0.0 and default is 1.0. **Tangent Scale Factor (SFTPNLT)** value ***Augment PenFactor (PENALT)** value is used in analysis. Writes SFTPNLT and value.

Max DDU (DDULMT)

Maximum value of DDU in a Newton-Raphson cycle. Value must be greater than or equal to zero. Default is 0.0, which means no limitation. If other value is enters, writes DDULMT and value.

Sticking Augment (TAUGMNT)

Augmentation for the sticking part of friction when **Contact Method** (METHOD) is set to **1..SEGTOSEG**. When off, which is the default, writes nothing and there is no augmentation. When on, writes TAUGMENT and a value of `1`, meaning use augmentation.

Infinitesimal Sliding (LINCNT)

Flag for Linear Contact (i.e., under infinitesimal assumption, small sliding with small deformation and rotation). When off, which is the default, nothing is written, meaning general contact. When on, writes LINCNT and value of `1`, which activates linear contact so the contact force is distributed based upon undeformed geometry.

Friction

The items in this section are written to the BCPARA entry.

Friction (FTYPE)

Friction type. When set to **0..None**, which is the default, writes nothing. When set to **6..Bilinear Coulomb**, writes FTYPE and a value of `6`, meaning Bilinear Coulomb friction. When set to **7..Bilinear Shear**, writes FTYPE and value of `7`, meaning Bilinear Shear friction.

Bilin (RVCNST)

Slip threshold for the bilinear friction model. Must be greater than or equal to 0.0. Default is 0.0. When set to a value other than 0.0, writes RVCNST and specified value.

Non Symmetric Matrix (SEGSYM)

Specifies if a symmetric or non-symmetric friction matrix will be used when **Contact Method** (METHOD) is set to **1..SEGTOSEG**. When off, which is the default, writes nothing, meaning symmetric friction matrix. When on, writes SEGSYM and a value of `1`, indicating a non-symmetric friction matrix will be used.

Cohesive Contact Options button

Displays the Cohesive Glued Contact Options dialog box, which contains options related to cohesive glued contact.

The **Enable Cohesive Contact (FGCFLG)** option must be enabled to use any options in this dialog box. When on, writes FGCFLG and a value of `1`, meaning Cohesive (Flexible) Glued Contact has been activated.

Normal Stiffness (FGCNST)

Equivalent normal contact stiffness of connector for Cohesive (Flexible) Glued Contact. Writes FGCNST and entered value.

Tangent Stiffness (FGCTST)

Equivalent tangential contact stiffness of connector for Cohesive (Flexible) Glued Contact. Writes FGCTST and entered value.

Normal Stress Function (FGCNSTR)

Select an existing function to define contact normal stress versus relative displacement for Cohesive (Flexible) Glued Contact. When a function is selected, writes FGCNSTR and the ID of the function.

Tangent Stress Function (FGCTSTR)

Select an existing function to define contact tangential stress versus relative displacement for Cohesive (Flexible) Glued Contact. When a function is selected, writes FGCTSTR and the ID of the function.

Normal Stress Function (FGCNST1)

Select an existing function to define contact normal stiffness versus rela-tive displacement or temperature for Cohesive (Flexible) Glued Contact. When a function is selected, writes FGCNST1 and the ID of the function.

Tangent Stress Function (FGCTST1)

Select an existing function to define contact tangential stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. When a function is selected, writes FGCTST1 and the ID of the function.

Reference Node (FGCRCEN)

ID of node on which the resultant contact and moment are based. Must be an existing node in the model. When not set to 0, writes FGCRCEN and ID of node.

Separation Control

The items in this section are written to the BCPARA entry.

Sep Control (ICSEP)

Select an option from the drop-down to control separation. Only used when **Contact Method** (METHOD) is set to **0..NODESURF**. Default is **0..Nodal Force > Separation Force**.

0..Nodal Force > Separation Force

The node separates and an iteration occurs if the force on the node is greater than the separation force. Writes nothing.

1..Prevent Separation Until Next Increment

If a node which was in contact at the end of the previous increment has a force greater than the separation force, the node does NOT separate in this increment, but separates at the beginning of the next increment. Writes ICSEP and value of 1.

2..Prevent New Node Separation

If a new node comes into contact during this increment, it is not allowed to separate during this increment, prevents chattering. Writes ICSEP and value of 2.

3..Both 1 and 2

Both 1 and 2 are in effect. Writes ICSEP and value of 3.

Separation (IBSEP)

Select an option from the drop-down to control if separation is based on stresses or forces. Default is **0..Force Based**.

0..Force Based

Separation based on forces. Writes nothing.

1..Absolute Stress Based

Separation based on absolute stresses (force/area). Writes IBSEP and value of 1.

2..Absolute Intg Pt Stress

Separation based on absolute stress (extrapolating integration point stresses). Writes IBSEP and value of 2.

3..Relative Nodal Stress

Relative nodal stress (force/area). Writes IBSEP and value of 3.

4..Relative Intg Pt Stress

Separation based on relative stress (extrapolating integration point stresses). Writes IBSEP and value of 4.

Max Sep (MAXSEP)

Maximum number of separations allowed in each increment. After the specified number of separations have occurred, if the standard convergence tolerance conditions are achieved, the step will converge. Default value is 9999. When a value other than 9999 is entered, writes MAXSEP and the value.

Sep Force/Stress (FNTOL)

Separation force (or stress if **Separation (IBSEP)** is set to an option other than **0..Force Based**) above which a node separates from a body. Automatically calculated when using a value of 0.0, which writes nothing. When a value other than 0.0 is entered, writes FNTOL and value.

Skip Sep Check (NODSEP)

Skip separation check in contact for any grid already touched and separated. The value indicates how times to skip within the current increment. Default is 5. When am integer value other than 5 is entered, writes NODSEP and value.

Simcenter Nastran Interface

DDAM

- Added read/write support for the SELMODE entry. This is specified using the **Mode Selection** section in the NASTRAN DDAM Solution Options dialog box in the **Analysis Set Manager**.

- Added read support for MPC Forces, Element Strains, and Membrane-only Stresses for Beams, Shells, and Laminates. These results are requested by PARAM,SHLMEM,YES.

A number of bugs were corrected.

For details, see *Section 8, "Analysis Program Interfaces"* in the Femap User Guide.

Nastran Interfaces (Simcenter and MSC/MD)

• Added read/write support for MODSEL (Simcenter Nastran) and a subset of the functionality of MODESE-LECT (MSC Nastran). This can be specified using the **Response Mode Selection** (a, b, c THRU d) section in the NASTRAN Modal Analysis dialog box in the **Analysis Set Manager** when creating an Analysis Set with **Analysis Type** set to **3..Transient Dynamic/Time History** or **4..Frequency/Harmonic Response**.

A number of bugs were corrected.

For details, see *Section 8, "Analysis Program Interfaces"* in the Femap User Guide.

ABAQUS Interface

• Added support to maintain the case of names during import of an input file.

• Added support to remove the leading characters of names during export, thus maintaining the names of entities entered by the user.

A number of bugs were corrected.

For details, see *Section 8, "Analysis Program Interfaces"* in the Femap User Guide.

Tools

• Added **Interference Factor**/**Proximity Factor** options to the **Tools → Check → Mesh Interference** command. In addition, consolidated the user interface to use a single dialog box instead of a series of questions.

First, select the elements to be checked, then specify options using the Check Mesh Interference dialog box.
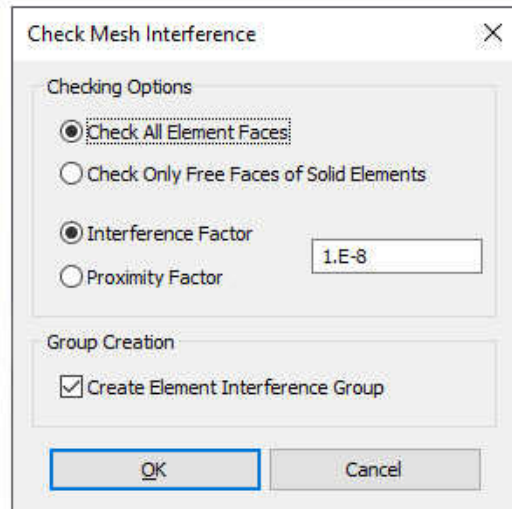
When **Check All Element Faces** is selected in the **Checking Options** section, all faces on solid elements are considered for interference checking. When **Check Only Free Faces of Solid Elements** is selected, only the free faces of solid elements are considered.

When **Interference Factor** is selected in the **Checking Options** section, an element has to overlap at least one other element by more than the specified value for the elements to be considered interfering. When **Proximity Factor** is selected, it acts in an opposite manner to **Interference Factor**, meaning if a gap smaller than the specified value exists between and element and any other element, those elements are considered interfering.

When **Create Element Interference Group** is on in the **Group Creation** section, a group of all elements which interfere with at least one other element is automatically created. If disabled, the elements are simply highlighted in the graphics window. If there no elements interfere, a group will not be created, even if this option is enabled.

OLE/COM API

New and modified API Objects and Attributes

- Added AnalysisStep (feAnalysisStep) object to the API.

Added StepTitle, StepLabel, StepID, InternalStepID, StepAnalysisType, UseGlobalStep, GlobalStepID, CaseID, SkipStandard, NasStepAlternateTextLocation, StartText, and EndText attributes to specify options in the MSC Nastran SOL400 Analysis Step dialog box.

Added NasInlStepOptOn, NasInlStepOptSkipNLSTEP, NasInlStepOptID, NasInlStepOptTOTTIM, NasInlStepOptCTRLDEF, NasInlStepOptMAXITER, NasInlStepOptMINITER, NasInlStepOptMAXBIS, NasInlStepOptCREEP, NasInlStepOptLoadStepping, NasInlStepOptNINC, NasInlStepOptNO, NasInlStepOptDTINITF, NasInlStepOptDTMINF, NasInlStepOptDTMAXF, NasInlStepOptNDESIR, NasInlStepOptSFACT, NasInlStepOptINTOUT, NasInlStepOptNSMAX, NasInlStepOptIDAMP, NasInlStepOptDAMP, NasInlStepOptEnableCriteria, NasInlStepOptIPHYS, NasInlStepOptLIMTAR, NasInlStepOptRSMALL, NasInlStepOptRBIG,

NasInlStepOptIncrements, NasInlStepOptGroupIDs, NasInlStepOptYT1_i, NasInlStepOptYT2_i, NasInlStepOp-tYT3_i, NasInlStepOptYT4_i, NasInlStepOptMT1_i, NasInlStepOptMT2_i, NasInlStepOptMT3_i, NasInlSte-pOptMT4_i, NasInlStepOptADJUST,NasInlStepOptMSTEP, NasInlStepOptRB, NasInlStepOptUTOL, NasInlStepOptTYPE, NasInlStepOptDTINITFA, NasInlStepOptMINALR, NasInlStepOptMAXALR, NasInlSte-pOptSCALEA, NasInlStepOptNDESIRA, NasInlStepOptNSMAXA, NasInlStepOptMECH, NasInlStepOpt-ConvU, NasInlStepOptConvP, NasInlStepOptConvW, NasInlStepOptConvV, NasInlStepOptConvN, NasInlStepOptConvA, NasInlStepOptEPSU, NasInlStepOptEPSP, NasInlStepOptEPSW, NasInlStepOptK-METHOD, NasInlStepOptKSTEP, NasInlStepOptMRCONV, NasInlStepOptMAXQN, NasInlStepOptMAXLS, NasInlStepOptLSTOL, and NasInlStepOptFSTRESS attributes to specify options in the various dialog boxes for MSC Nastran SOL 400 Control Parameters in an Analysis Step.

Added NasModeOn, NasModeEigrSet, NasModeMethod, NasModeSolutionType, NasModeEstRoots, NasMod-eDesiredRoots, NasModeNormOpt, NasModeNormNode, NasModeNormDOF, NasModeMassForm, vNasMode-FreqRange, NasModesSkipEigr, vNasModeImagFreqRange, NasModesComplexConverge, NasModesComplexRegionWidth, and NasModeDampOverall in the NASTRAN Modal Analysis dialog box for an Analysis Step.

Added BCSet, vBCSet, ContactOption, and ContactSetType attributes to specify options in the Boundary Condi-tions dialog box for an Analysis Step.

Added Output, vOutput, Echo, Destination, Imaginary, CornerOutput, IntegrationOutput, NasOmodesOn, NasO-modes, and NasCurvatureStrain attributes to specify options in the NASTRAN Output Requests dialog box for an Analysis Step.

- Added GlobalStep (feGlobalStep) object to the API.

Added GlobalStepTitle, GlobalStepLabel, globalstepID, GlobalStepAnalysisType, SkipStandard, NasGlobalAlter-nateTextLocation, StartText, and EndText attributes to specify options in the MSC Nastran SOL400 Global Analysis Step dialog box.

Added NasInlStepOptOn, NasInlStepOptSkipNLSTEP, NasInlStepOptID, NasInlStepOptTOTTIM, NasInlSte-pOptCTRLDEF, NasInlStepOptMAXITER, NasInlStepOptMINITER, NasInlStepOptMAXBIS, NasInlStepOpt-CREEP, NasInlStepOptLoadStepping, NasInlStepOptNINC, NasInlStepOptNO, NasInlStepOptDTINITF, NasInlStepOptDTMINF, NasInlStepOptDTMAXF, NasInlStepOptNDESIR, NasInlStepOptSFACT, NasInlSte-pOptINTOUT, NasInlStepOptNSMAX, NasInlStepOptIDAMP, NasInlStepOptDAMP, NasInlStepOptEnableCri-teria, NasInlStepOptIPHYS, NasInlStepOptLIMTAR, NasInlStepOptRSMALL, NasInlStepOptRBIG, NasInlStepOptIncrements, NasInlStepOptGroupIDs, NasInlStepOptYT1_i, NasInlStepOptYT2_i, NasInlStepOp-tYT3_i, NasInlStepOptYT4_i, NasInlStepOptMT1_i, NasInlStepOptMT2_i, NasInlStepOptMT3_i, NasInlSte-pOptMT4_i, NasInlStepOptADJUST,NasInlStepOptMSTEP, NasInlStepOptRB, NasInlStepOptUTOL, NasInlStepOptTYPE, NasInlStepOptDTINITFA, NasInlStepOptMINALR, NasInlStepOptMAXALR, NasInlSte-pOptSCALEA, NasInlStepOptNDESIRA, NasInlStepOptNSMAXA, NasInlStepOptMECH, NasInlStepOpt-ConvU, NasInlStepOptConvP, NasInlStepOptConvW, NasInlStepOptConvV, NasInlStepOptConvN, NasInlStepOptConvA, NasInlStepOptEPSU, NasInlStepOptEPSP, NasInlStepOptEPSW, NasInlStepOptK-METHOD, NasInlStepOptKSTEP, NasInlStepOptMRCONV, NasInlStepOptMAXQN, NasInlStepOptMAXLS, NasInlStepOptLSTOL, and NasInlStepOptFSTRESS attributes to specify options in the various dialog boxes for MSC Nastran SOL 400 Control Parameters in a Global Step.

Added NasModeOn, NasModeEigrSet, NasModeMethod, NasModeSolutionType, NasModeEstRoots, NasMod-eDesiredRoots, NasModeNormOpt, NasModeNormNode, NasModeNormDOF, NasModeMassForm, vNasMode-FreqRange, NasModesSkipEigr, vNasModeImagFreqRange, NasModesComplexConverge, NasModesComplexRegionWidth, and NasModeDampOverall in the NASTRAN Modal Analysis dialog box for a Global Step.

Added BCSet, vBCSet, ContactOption, and ContactSetType attributes to specify options in the Boundary Condi-tions dialog box for a Global Step.

Added Output, vOutput, Echo, Destination, Imaginary, CornerOutput, IntegrationOutput, NasOmodesOn, NasO-modes, and NasCurvatureStrain attributes to specify options in the NASTRAN Output Requests dialog box for a Global Step.

- Added PublishTool (fePublishTool) object to the API. In addition, added PublishTable, which can be used by the PublishTable object to the API. PublishTool and PublishTable have no attributes, only methods.

- Added NasInlStepOptOn, NasInlStepOptSkipNLSTEP, NasInlStepOptID, NasInlStepOptTOTTIM, NasInlSte-pOptCTRLDEF, NasInlStepOptMAXITER, NasInlStepOptMINITER, NasInlStepOptMAXBIS, NasInlSte-pOptCREEP, NasInlStepOptLoadStepping, NasInlStepOptNINC, NasInlStepOptNO, NasInlStepOptDTINITF, NasInlStepOptDTMINF, NasInlStepOptDTMAXF, NasInlStepOptNDESIR, NasInlStepOptSFACT, NasInlSte-pOptINTOUT, NasInlStepOptNSMAX, NasInlStepOptIDAMP, NasInlStepOptDAMP, NasInlStepOptEna-bleCriteria, NasInlStepOptIPHYS, NasInlStepOptLIMTAR, NasInlStepOptRSMALL, NasInlStepOptRBIG, NasInlStepOptIncrements, NasInlStepOptGroupIDs, NasInlStepOptYT1_i, NasInlStepOptYT2_i, NasInlSte-pOptYT3_i, NasInlStepOptYT4_i, NasInlStepOptMT1_i, NasInlStepOptMT2_i, NasInlStepOptMT3_i, NasIn-lStepOptMT4_i, NasInlStepOptADJUST,NasInlStepOptMSTEP, NasInlStepOptRB, NasInlStepOptUTOL, NasInlStepOptTYPE, NasInlStepOptDTINITFA, NasInlStepOptMINALR, NasInlStepOptMAXALR, NasInl-StepOptSCALEA, NasInlStepOptNDESIRA, NasInlStepOptNSMAXA, NasInlStepOptMECH, NasInlSte-pOptConvU, NasInlStepOptConvP, NasInlStepOptConvW, NasInlStepOptConvV, NasInlStepOptConvN, NasInlStepOptConvA, NasInlStepOptEPSU, NasInlStepOptEPSP, NasInlStepOptEPSW, NasInlStepOptK-METHOD, NasInlStepOptKSTEP, NasInlStepOptMRCONV, NasInlStepOptMAXQN, NasInlStepOpt-MAXLS, NasInlStepOptLSTOL, and NasInlStepOptFSTRESS attributes to specify options in the various dialog boxes for MSC Nastran SOL 400 Control Parameters via an Analysis Case Object.

- Added a number of attributes to the Analysis Manager Object.

Added NasModeSelOn, NasModeSelInc, and NasModeSel attributes to specify options in the NASTRAN Modal Analysis dialog box via an Analysis Manager Object.

Added NasDdamSelModeMethod and NasDdamSelModeVal attributes to specify options in the NASTRAN DDAM Solution Options dialog box via an Analysis Manager Object.

Added NasNonlinBehaviorOn, NasNonlinBehaviorCreepExp, NasNonlinBehaviorEnable, NasNonlinBehavior-CreepType, NasNonlinBehaviorCreepImp, NasNonlinBehaviorASSM, NasNonlinBehaviorTSHEAR, NasNonlin-BehaviorLRGSTRN, NasNonlinBehaviorPROPMAP, NasNonlinBehaviorPROPBEH, NasNonlinBehaviorDIRECT, NasNonlinBehaviorTHICKOP, NasNonlinBehaviorIPRINT, NasNonlinBehavior-VALD1, NasNonlinBehaviorVALD2,

NasNonlinBehaviornINLAM, NasNonlinBehaviorCreepOn, NasNonlinBe-haviorCPROJ attributes to specify options in the Nonlinear Behavior Options dialog box via an Analysis Manager Object.

Added NasAutoCtOn, NasAutoCtEnabled, NasAutoCtBodyVal, NasAutoCtCTYPE, NasAutoCtDISTANCE, NasAutoCtESET, NasAutoCtSeedESet, NasAutoCtSeedGSet, NasAutoCtPropSet, NasAutoCtRigidSet, NasAuto-CtBEAMCNT, NasAutoCtBEAMCAP, NasAutoCt2D, NasAutoCtEDGECNT, NasAutoCtFTYPE, NasAutoCt-FRIC, NasAutoCtFRICTABLE, NasAutoCtFRICVal, NasAutoCtIGNTHK, NasAutoCtINISTF, NasAutoCtSelfCont, NasAutoCtRegion, and NasAutoCtConProp attributes to specify options in the Automatic Contact Generation dialog box via an Analysis Manager Object.

Added NasMscCtTHKOFF, NasMscCtLINCNT, NasMscCtSFNPNLT, NasMscCtSFTPNLT, NasMscCtFGCFLG, NasMscCtFGCNST, NasMscCtFGCTST, NasMscCtFGCNSTR, NasMscCtFGCTSTR, NasMscCtFGCNST1, NasMscCtFGCTST1, and NasMscCtFGCRCEN attributes to specify options in the MSC Nastran Contact Solver Parameters dialog box via an Analysis Manager Object.

Added NasInlStepOptOn, NasInlStepOptSkipNLSTEP, NasInlStepOptID, NasInlStepOptTOTTIM, NasInlSte-pOptCTRLDEF, NasInlStepOptMAXITER, NasInlStepOptMINITER, NasInlStepOptMAXBIS, NasInlStepOpt-CREEP, NasInlStepOptLoadStepping, NasInlStepOptNINC, NasInlStepOptNO, NasInlStepOptDTINITF, NasInlStepOptDTMINF, NasInlStepOptDTMAXF, NasInlStepOptNDESIR, NasInlStepOptSFACT, NasInlSte-pOptINTOUT, NasInlStepOptNSMAX, NasInlStepOptIDAMP, NasInlStepOptDAMP, NasInlStepOptEnableCri-teria, NasInlStepOptIPHYS, NasInlStepOptLIMTAR, NasInlStepOptRSMALL, NasInlStepOptRBIG, NasInlStepOptIncrements, NasInlStepOptGroupIDs, NasInlStepOptYT1_i, NasInlStepOptYT2_i, NasInlStepOp-tYT3_i, NasInlStepOptYT4_i, NasInlStepOptMT1_i, NasInlStepOptMT2_i, NasInlStepOptMT3_i, NasInlSte-pOptMT4_i, NasInlStepOptADJUST,NasInlStepOptMSTEP, NasInlStepOptRB, NasInlStepOptUTOL, NasInlStepOptTYPE, NasInlStepOptDTINITFA, NasInlStepOptMINALR, NasInlStepOptMAXALR, NasInlSte-pOptSCALEA, NasInlStepOptNDESIRA, NasInlStepOptNSMAXA, NasInlStepOptMECH, NasInlStepOpt-ConvU, NasInlStepOptConvP, NasInlStepOptConvW, NasInlStepOptConvV, NasInlStepOptConvN, NasInlStepOptConvA, NasInlStepOptEPSU, NasInlStepOptEPSP, NasInlStepOptEPSW, NasInlStepOptK-METHOD, NasInlStepOptKSTEP, NasInlStepOptMRCONV, NasInlStepOptMAXQN, NasInlStepOptMAXLS, NasInlStepOptLSTOL, and NasInlStepOptFSTRESS attributes to specify options in the various dialog boxes for MSC Nastran SOL 400 Control Parameters in the Master Case of an Analysis Set via Analysis Manager Object.

- Added SurfaceContactFlag, FrictionType, FrictionValue, FrictionTable, SmoothingControl, ContactCheck, MidNodeProjection, DiscontinuityValue, ControlNode, MotControlType, RotControlNode, AngVelFlag, AngVelTable, AngVelValue, RotAxisFlag, RotAxisTable, RotAxisValue, MotionFlag, MotionTable, Motion-Value, AppAngVelVal, AppRotAxisVal, AppVelVal, GrowthComp, and GrowthTable attributes to the Con-nection Region Object.

- Added ContourElemBreakLinearParabolic attribute to the View Object.

New and Updated API Methods

- Added InitAnalysisStep, DeleteAnalysisStep, and ActiveContactEntities methods to the AnalysisStep Object.

- Added InitGlobalStep, DeleteGlobalStep, and ActiveContactEntities methods to the GlobalStep Object.

- Added Clear, SetFormat, SetParagraphIndent, SetParagraphAlignment, SetParagraphSpacing, AddStyle, AddParagraph, AddPicture, AddView, AddDataTable, AddFunctionTableEditor, AddEntityEditor, AddMes-sages, AddChart, UserTable, PublishToClipboard, and PublishToFile methods to the PublishTool Object. In addition, added AddColumn, AddColumnGroupTitle, AddRowNumbering, SetRealFormat,SetTableStyle, SetRowTitle, SetCellStyle, SetCellReal, SetCellInteger, SetCellText, SetMultiCellStyle, SetMultiCellReal, SetMultiCellInteger, SetMultiCellText, and Publish methods for PublishTable.

- Added IsNotEmpty, HasOne, and HasMultiple methods to the Set Object.

- Added GetStep, PutStep, and DeleteStep for MSC Nastran SOL 400 to Analysis Case Object.

- Added RemoveDuplicates method to the Connection Region Object.

- Added IsFreeEdge and IsOnSolidRegion methods to the Curve Object.

The following functions have been added or updated:

- feAppLoadColorPalette

- feAppLoadContourPalette

- feGroupCondense

- feMeshSizeCurve2

- feMeshSizePropagate

- feSurfaceMidAuto4

- feCheckElemInterference2

New and updated Global Variables

- Added Pref_NewPick to set the New Picking preference on the Graphics tab of File, Preferences.

- Added Pref_Filter_Analysis to set the Enable Analysis Filtering option on the Interfaces tab of File, Prefer-ences. In addition, added Pref_Filter_SC_Nastran_Linear, Pref_Filter_SC_Nastran_MSNL, Pref_Fil-ter_SC_Nastran_MSNLK, Pref_Filter_SC_Nastran_ADVNL, Pref_Filter_SC_Nastran_ADVNLE, Pref_Filter_MSC_Nastran, Pref_Filter_Autodesk_Nastran, Pref_Filter_ABAQUS, Pref_Filter_ANSYS, Pref_-Filter_DYNA, Pref_Filter_SINDA, and Pref_Filter_MARC to control the individual options in the Select Solv-ers to Filter dialog box.

- Added Info_MeshSizePropagation_On to enable the Propagate Sizing option in the dialog boxes for various Mesh Sizing commands. In addition, added Info_MeshSizePropagation_SelMode, Info_MeshSizePropaga-tion_SlavedMapped, Info_MeshSizePropagation_EdgeOption,

Info_MeshSizePropagation_SameSolid, Info_MeshSizePropagation_UseTolerance, and Info_MeshSizePropagation_Tolerance to control the options in the Mesh Sizing Propagation Options dialog box.

• Added Pref_API_UseAlternateFileDialog and Pref_API_HonorWorkingDirectory to control specify options needed for improved access for Teamcenter Simulation (TC Sim).

Preferences

Graphics

• Added **New Picking** to the **Graphics Options** section.

When on, which is the default, **New Picking** uses new picking introduced in version 2021.1 which takes advantage of *Open MP* to improve performance when selecting a large number of entities from the graphics window, especially when using area picking (Box, Circle, Polygon, or Freehand) and/or Front Pick. Unless there is a problem picking entities, **New Picking** should remain on. **Fast Picking** uses older picking methods, but for the time being should also remain on. Eventually, **New Picking** will become the only available option and these preferences will be removed.

• Updated **OpenMP** in the **Advanced/Debug Options** section. To be activated by default. In addition, now set a default value of 4 for **Number of Threads**.

If any issues are encountered when using this option, please deactivate this option, and then contact Customer Support to describe the issue.

This option, which is on by default, activates **OpenMP** functionality (i.e. multi-threading). Once activated, click the **Options...** button to set **Number of Threads**, which should not exceed the number of cores available on the machine running Femap. The default for **Number of Threads** is 4.

This functionality has been implemented to improve performance for a limited number of commands and operations including:

___ Picking done when the **New Picking** option is activated.
___ When using the **View → Advanced Post → Dynamic IsoSurface** and **View → Advanced Post → Cynamic Cutting Plane** commands.
___ Placing a contour plot on certain types of entities when post-processing

Interfaces

• Added an **Enable Analysis Filtering** option and a **Select Analysis Filters** icon button, which accesses the Select Solvers to Filter dialog box.

When activated, this option is used in conjunction with the solvers selected using the **Select Analysis Filters** icon button to limit solver-specific options found in various locations throughout Femap, including dialog boxes and dockable panes.
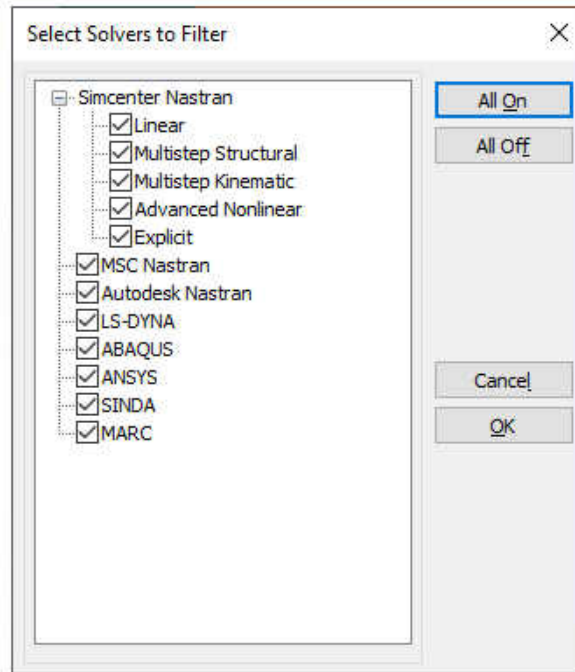
Solvers which can be filtered are MSC Nastran, Autodesk Nastran, LS-DYNA, ABAQUS, ANSYS, SINDA, and MARC. For Simcenter Nastran, certain analysis types can be filtered, including Linear (Contact options including glued contact for most solution sequences), Multistep Structural (SOL 401), Multistep Kinematic (SOL 402), Advanced Nonlinear (SOL 601), and/or Explicit (SOL 701).

The **All Off** button can be used to quickly select all options, while **All Off** can quickly deselects all options.