20.1

# What’s New in FEMAP

FEMAP 2020.1 included enhancements and new features, which are detailed below:

VERY IMPORTANT - Output Vector ID Ranges Have Changed!

User Interface

Geometry

Meshing

Elements

Loads and Constraints

Simulation Entities - New for 2020.1!

Connections (Regions, Properties, and Connectors)

Aeroelasticity

Listing

Groups and Layers

Views

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

Tools

Model Merge

OLE/COM API

Preferences

FEMAP 2019.1 MP 1 included enhancements and new features, which are detailed below:

Geometry Interfaces

Analysis Program Interfaces

FEMAP 2019.1 included enhancements and new features, which are detailed below:

User Interface

Geometry

Meshing

Elements

Properties

Layups

Loads and Constraints

Connections (Regions, Properties, and Connectors)

Optimization

Aeroelasticity

Listing

Groups and Layers

Views

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

Tools

Libraries

OLE/COM API

Preferences

FEMAP 12.0 and 12.0.1 includes enhancements and new features, which are detailed below:

User Interface

Geometry

Meshing

Elements

Materials

Properties

Layups

Loads and Constraints

Connections (Regions, Properties, and Connectors)

Optimization

Listing

Groups and Layers

Views

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

Tools

Model Merge

OLE/COM API

Preferences

# What's New - version 2020.1

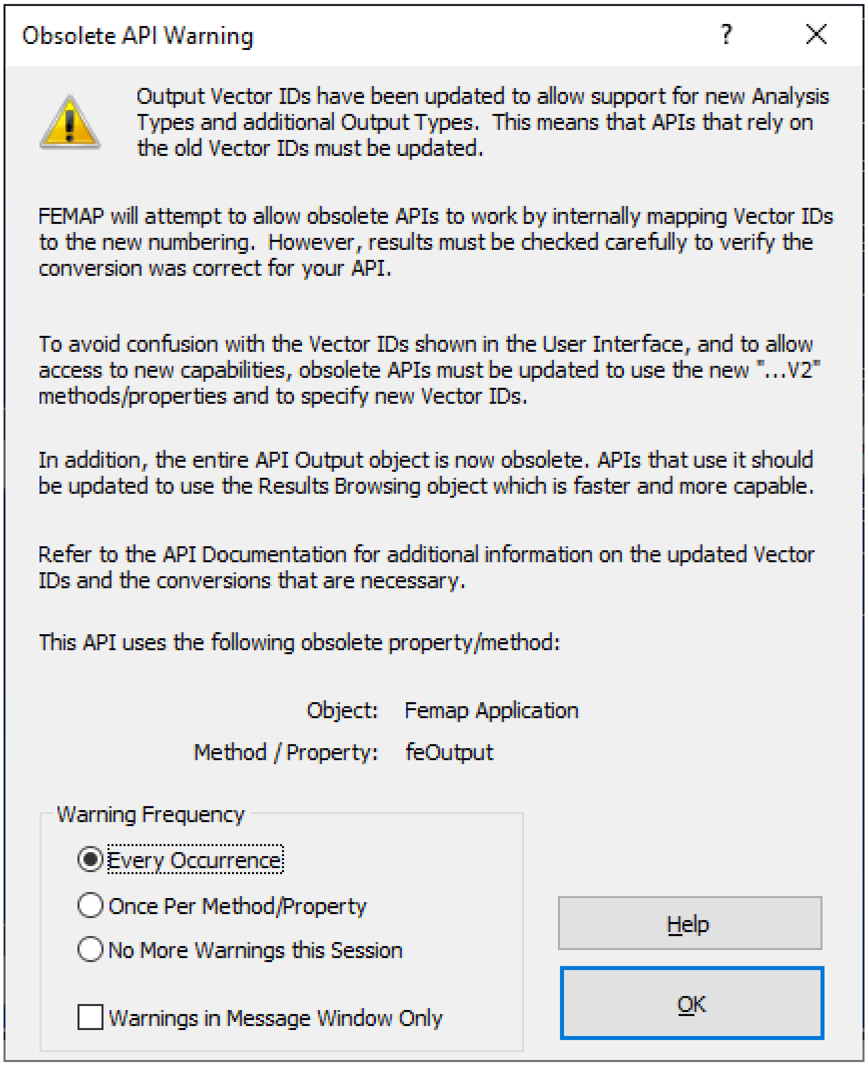
## VERY IMPORTANT - Output Vector ID Ranges Have Changed!

- The static ranges of Output Vector IDs in previous versions of FEMAP cannot accommodate the large numbers of new output quantities and element types that are found in analyses being run by a large number of FEMAP users. To accommodate the ongoing addition of output quantities, some of the existing output vectors have been moved to new ID ranges. If opening a model or importing a neutral file from a version of FEMAP prior to 2020.1, the output vector IDs will be updated automatically and there should be no difference in behavior. The following table shows both the old and new output vector ID ranges. For information about this topic, see Section 2.4, "Working with Output in FEMAP 2020.1 and above" in the *FEMAP API Reference*.

| Output Vector Range Comparison | | |
|---|---|---|
| **Output Type** | **Vector Ranges from 9.3 through 2019.1** | **Vector Range 2020.1 and Above** |
| Nodal output | 1 to 2,999 | 1 to 2,999 |
| Line element output | 3,000 to 5,999 | 3,000 to 5,999 |
| Plate element output | 6,000 to 7,599 | 6,000 to 9,999 |
| Plate corner output | 100,000 to 299,999 | 10,000 to 25,999 |
| Not currently used | 7,600 to 59,999 | 26,000 to 59,999 |
| Solid element output | 60,000 to 69,999 | 60,000 to 69,999 |
| Solid corner output | 70,000 to 73,999 | 70,000 to 79,999 |
| Output on any element type | 80,000 to 89,999 | 80,000 to 89,999 |
| PATRAN elemental output | 90,000 to 99,999 | 90,000 to 99,999 |
| Not currently used | 300,000 to 999,999 | 100,000 to 999,999 |
| Plate Laminate Ply output | 1,000,000 to 1,999,999 | 1,000,000 to 3,999,999 |
| Plate Laminate Ply corner | 2,000,000 to 5,999,999 | 4,000,000 to 15,999,999 |
| Solid Laminate Ply output | 6,000,000 to 6,499,999 | 16,000,000 to 17,499,999 |
| Solid Laminate Ply corner | 6,500,000 to 8,499,999 | 17,500,000 to 23,499,999 |
| User Defined output | 9,000,000 to 9,999,999 | 24,000,000 to 24,999,999 |
| Complex Phase output | Add 10,000,000 to above ranges | Add 25,000,000 to above ranges |
| Complex Real output | Add 20,000,000 to above ranges | Add 50,000,000 to above ranges |
| Complex Imaginary output | Add 30,000,000 to above ranges | Add 75,000,000 to above ranges |

- Perhaps more importantly, for anyone who has developed and/or used scripts and/or full applications which use the Output Object in the FEMAP API, this Object has been FULLY DEPRECATED. Moving forward, everything related to output when using the API MUST be done with the Results Browsing Object. In addition, a ResultsVectorIDQuery Object has been added in an attempt to not require recoding of API programs in the future. For information on the changes, see OLE/COM API section.

- If using an API script or program which contains deprecated calls, methods, or properties regarding output vector IDs or the Output Object, the *Obsolete API Warning* dialog box will appear, which has options for the frequency of warnings and/or how the warning messages will be issued.




The *Warning Frequency* section can be used to control how often FEMAP warns the user about using deprecated API calls and/or how the user will be warned for a given FEMAP session:

***Every Occurrence*** - This is the default setting and issues a warning message every time a deprecated call, method, and/or property involving output vector IDs or the Output Object (feOutput) is encountered in an API script or program.

***Once Per Method/Property*** - When selected, a warning will be issued a single time for each deprecated for each call, method, or property involving output vector IDs or the Output Object (feOutput) is encountered in an API script or program.

***No More Warnings this Session*** - When selected, no more warning messages will be issued while the current FEMAP session is open for any deprecated call, method, and/or property involving output vector IDs or the Output Object (feOutput) is encountered in an API script or program.

When enabled, the *Warnings in Message Window Only* option will no longer display the *Obsolete API Warning* dialog box and only list these warning messages to the *Messages* window until the FEMAP session is closed.

Once FEMAP is closed, all settings will return to the default settings and the next time FEMAP is opened, the user can select their options for deprecated API calls again.

- Any Program File which has hard-coded Output Vector IDs, will need to be manually updated.

# User Interface

General, Menu, Toolbars, Model Info tree, Meshing Toolbox, and Function/Table Editor

## General

- Updated FEMAP to support “Large IDs” (i.e., IDs larger than 99,999,999). Not all solvers will support values higher than 99,999,999, so it is up to the user to determine is using “Large IDs” is appropriate.
- Updated all icons throughout the User Interface. This includes icons in the menu structure, on toolbars, in dialog boxes, and used in dockable panes.
- Updated all “Palette” buttons with “color block” buttons which now show the color of the entity, including settings for *Transparency*. In addition, if any setting for *Line Style* is specified, it will also be shown below the “color block” in the button.
- Added *Select from Standard Select Dialog* icon to dialog boxes which bring up the “multi-select dialog box with check boxes” by default (for example, *List, Model, Load Definition* or *Delete, Output, Set*). This can be helpful if the IDs for entities that need to be selected already exist in a list or spreadsheet, as the *Paste* functionality on the *Pick^* menu can then be used.

## Menu

- Added *Tools, Check, Mesh Interference* to the *Tools* menu. See Tools section for more information.
- Added *Model, Simulation Entities, Monitor Points* and *Model, Simulation Entities, Direct Matrix Input* to the *Model* menu. See Simulation Entities - New for 2020.1! section for more information. Also, added *Model, Load, Elemental on Face* command to Model menu. See Loads and Constraints section for more information.
- Added *Modify, Renumber, Table* and *Modify, Renumber, Data Surface* commands to the *Modify* menu, which are used to renumber Tables or Data Surfaces, respectively, by *Original ID*, *Selection Order*, or *Type*.
- Added *Delete, Model, Monitor Points* and *Delete, Model, Direct Matrix Input* to the *Delete* menu, which are simply used to delete Monitor Points and Direct Matrix Input entities, respectively.

## Toolbars

- Added *Nodal on Face* and *Elemental on Face* commands to the *Loads* toolbar, which will launch the *Model, Load, Nodal on Face* or *Model, Load, Elemental on Face* commands, respectively.
- Added *Between Geometry* command to *Lines* toolbar, which launches the *Geometry, Line, Between Geometry* command.

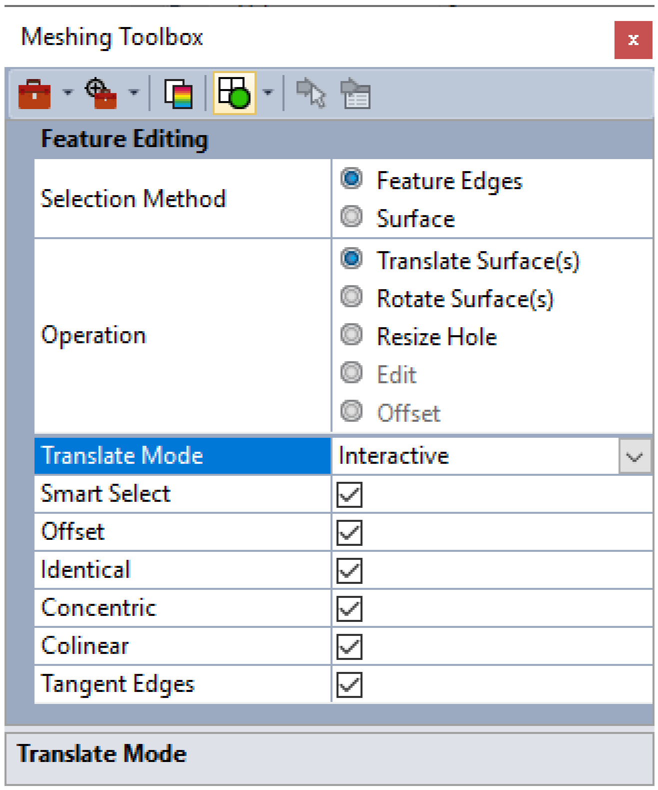
## Model Info tree

- Added *Simulation Entities* branch and underlying branches for *Monitor Points* and *Matrix Inputs*, which allow 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.
- Added Visibility check boxes (on/off) for *Monitor Points*. There is also a context-sensitive menu for the Visibility check boxes which offers *Show Selected Only*, *Show Selected*, *Hide Selected*, *Select Show Only...*, *Select to Hide...*, *Show All*, *Hide All*, and *Show/Hide Reverse* commands.
- Added *Tables* branch under the *Model* branch, which allows a table entity to be reloaded/edited via the *Function/Table Editor* or highlighted table(s) to be deleted or renumbered.
- Added *Elemental on Face* command to the context-sensitive menu for *Load Definitions*, which will launch the *Model, Load, Elemental on Face* command.
- Added *Renumber* command to the context-sensitive menu for *Data Surfaces*, which renumbers the highlighted data surface(s).
- Enhanced functionality of any command which creates a new Load Set or Constraint Set to automatically “expand” the newly created Set in the *Model Info* tree. This only occurs when the *Model Tree* is visible.

- Enhanced *Copy to Set* command on the context-sensitive menus for *Load Definition* to allow the highlighted Load Definition(s) to be copied into multiple Load Sets. In addition, improved the *Copy to Set* command on the context-sensitive menu for *Constraint Definition* in a similar manner.

## Meshing Toolbox

- *Feature Editing* tool - Added *Smart Select* option, along with *Offset*, *Identical*, *Concentric*, *Colinear*, and *Tangent Edges* options, to control automatic addition of surfaces to a set of already selected surfaces for more robust feature editing. In addition, when *Operation* is set to *Translate Surface(s),* added the *Translate Mode* drop-down, which offers multiple workflow options.




The *Smart Select* option is designed to automatically identify and select additional surfaces of a solid model to make feature editing easier and more robust. When *Smart Select* is enabled, five additional options, *Offset*, *Identical*, *Concentric*, *Colinear*, and/or *Tangent Edges*, will appear in the *Feature Editing* tool. Each of these options corresponds to a “selection rule”, which is used to automatically add additional surfaces to the selected surface(s):

***Offset*** - adds surfaces that are true offsets of the selected surface(s).

***Identical*** - adds surfaces which share the same underlying plane, cylinder, cone, torus, or sphere as the selected surface(s).

***Concentric*** - adds surfaces which are concentric cylinders, cones, and torii to the selected surface(s).

***Colinear*** - adds surfaces which share the same cylindrical, conical, toroidal, or spherical axis of the selected surface(s)

***Tangent Edges*** - adds surfaces which are connected to the selected surface(s) via a tangent edge.

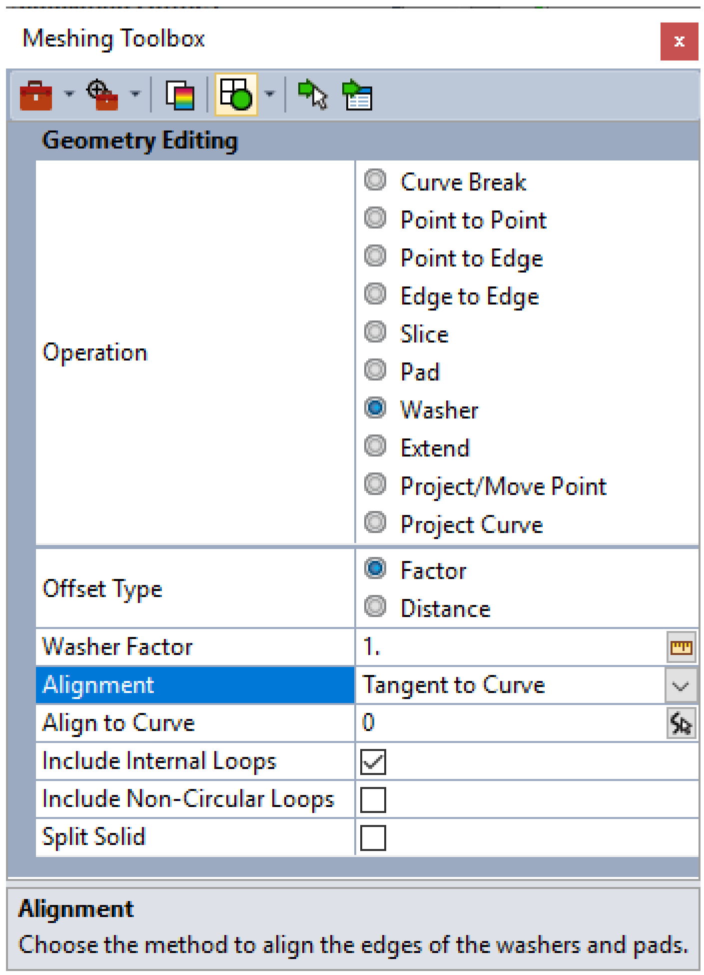
The *Translate Mode* drop-down only appears when *Operation* is set to *Translate Surface(s)*. It has three options:

***Default*** - First, use *Vector to Move Along* to specify a translation vector and, optionally, a *Distance*, then pick any number of surfaces or feature edges.

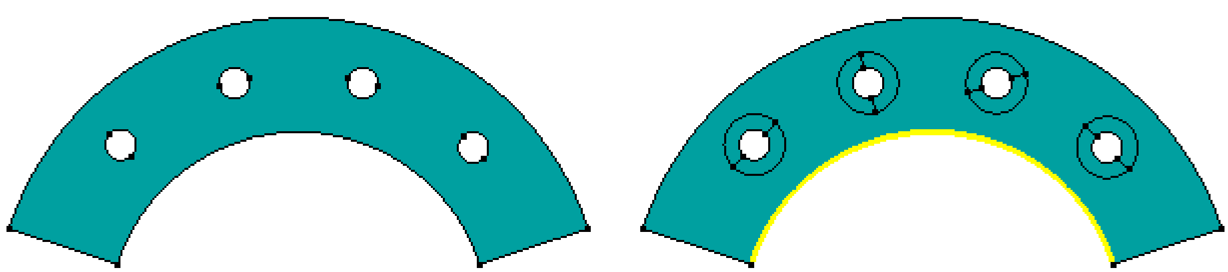
***Interactive*** - First, select any number of surfaces or feature edges, then specify a vector using the standard vector definition dialog box, then click *OK*.

***Auto Normal*** - Only available when *Select Method* is set to “Surface”. First, enter an offset distance in the *Value* field, then select any number of surfaces.

- *Feature Editing* tool - Added ability to translate an internal loop (i.e., hole, slot, cutout) within a surface when *Selection Method* is set to *Feature Edges* and *Operation* is set to *Translate Surface(s)*. In addition, added similar functionality to rotate an internal loop within a surface when *Operation* is set to *Rotate Surface(s)*.
- *Geometry Editing* tool - Added *Alignment* drop-down when *Operation* is set to *Washer*, which provides four options, “Automatic”, “Vector”, “Tangent to Curve”, and “Perpendicular to Curve”, which can be used to “align” the split lines of a washer to be parallel to a specified vector, tangent to selected curve, or perpendicular to a selected curve.




For example, picture on left is original geometry. In picture on right, *Alignment* option used, from left to right, are “Tangent to Curve” (highlighted curve), “Perpendicular to Curve” (highlighted curve), “Vector”, and “Automatic”.



## *Function/Table Editor*

- Added *Show Function in Charting* icon, which will automatically plot the function currently loaded in the *Function/Table Editor* to the *Charting* pane.

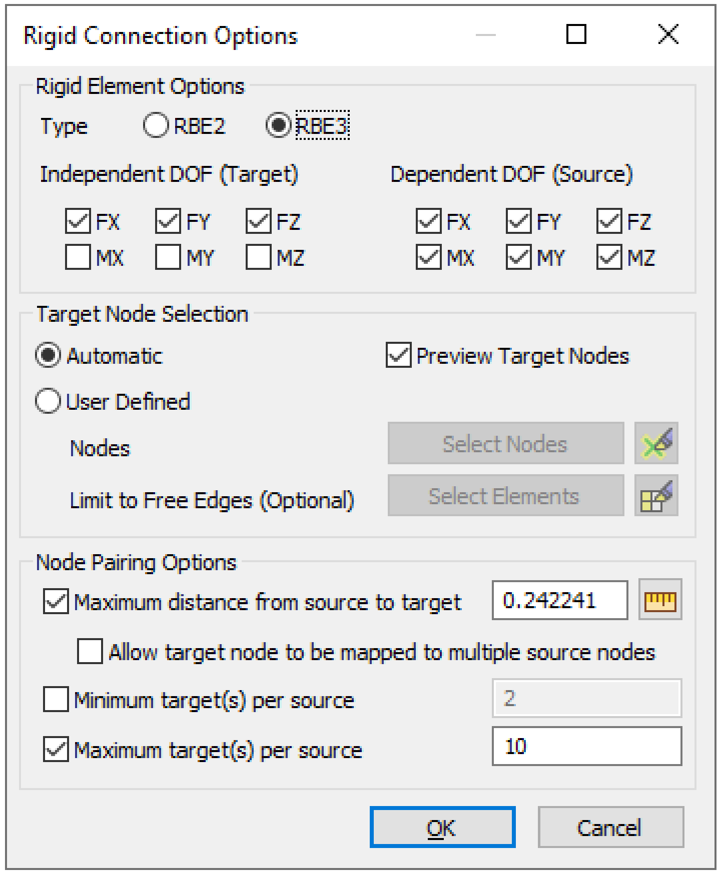
## Geometry

- Added *Coordinate Systems* option to *Other Entities to Include* section for commands on the *Geometry, Copy...* menu. In addition, created *Parameters* section, changed *Copy to Active Layer* to *Assign to Active Layer* and moved it into this section, along with the new *Assign Active Color…* and *Assign to Active CSys* options.
- Added *Coordinate Systems* option to *Other Entities to Include* section for commands on the *Geometry, Rotate...* menu. In addition, created *Parameters* section, changed *Copy to Active Layer* to *Assign to Active Layer* and moved it into this section, along with the new *Assign Active Color…* and *Assign to Active CSys* options.
- Added *Coordinate Systems* option to *Other Entities to Include* section for commands on the *Geometry, Reflect...* menu. In addition, created *Parameters* section, changed *Copy to Active Layer* to *Assign to Active Layer* and moved it into this section, along with the new *Assign Active Color…* and *Assign to Active CSys* options.
- Added *Coordinate Systems* option to *Other Entities to Include* section for geometry commands on the *Modify, Move By...*; *Modify, Rotate By...*; *Modify, Reflect...*; and *Modify, Align...* menus.
- Added *Alignment* options when creating Washers via the *Geometry Editing* tool in the *Meshing Toolbox*. See Meshing Toolbox section for more information.
- Added ability to translate an internal loop (i.e., hole, slot, cutout) within a surface, via the *Feature Editing* tool in the Meshing Toolbox, when *Selection Method* is set to *Feature Edges* and *Operation* is set to *Translate Surface(s).* In addition, added similar functionality to rotate an internal loop within a surface when *Operation* is set to *Rotate Surface(s)*. See Meshing Toolbox section for more information.
- Improved removal of fillets and blends, including “stepped” blends, when using commands and features of the Meshing Toolbox that are designed to perform this task.

## Meshing

- Added *Model Free Edges* option to the standard “Edge Selection” dialog box which is now used in a number of meshing commands. When this option is selected, any free edges of elements being considered by the command will automatically be selected.
- Added *Coordinate Systems* option to *Other Entities to Include* section for commands on the *Mesh, Copy, Node* and *Mesh, Copy, Element* commands. In addition, created *Parameters* section, changed *Copy to Active Layer* to *Assign to Active Layer* and moved it into this section, along with the new *Assign Active Color…* and *Assign to Active CSys* options.
- Added *Coordinate Systems* option to *Other Entities to Include* section for commands on the *Mesh, Rotate, Node* and *Mesh, Rotate, Element* commands. In addition, created *Parameters* section, changed *Copy to Active Layer* to *Assign to Active Layer* and moved it into this section, along with the new *Assign Active Color…* and *Assign to Active CSys* options.
- Added *Coordinate Systems* option to *Other Entities to Include* section for commands on the *Mesh, Reflect, Node* and *Mesh, Reflect, Element* commands. In addition, created *Parameters* section, changed *Copy to Active Layer* to *Assign to Active Layer* and moved it into this section, along with the new *Assign Active Color…* and *Assign to Active CSys* options.
- Added *Coordinate Systems* option to *Other Entities to Include* section for Coordinate System, Node, and Element commands on the *Modify, Move By...*; *Modify, Rotate By...*; *Modify, Reflect...*; and *Modify, Align...* menus.
- Updated *Mesh, Edge/Skin Elements, Planar Elements on Faces* command to display the standard “Face Selection” dialog box instead of having the user select elements, then faces, which was often an non-required step. To restrict which elements to “skin”, simply use the “From # Elements” button in the standard “Face Selection” dialog box to limit which elements to consider. In addition, updated *Mesh, Edge/Skin Elements, Line Elements on Edges* command in a similar manner, only the standard “Edge Selection” dialog box is displayed instead.
- Updated *Mesh, Geometry, Solids* command by adding the “2..Into Model” option to the *Merge Nodes* dropdown in the *Automesh Solids* dialog box. This differs from “1..New Nodes”, which would not merge nodes of newly created mesh with nodes of currently existing mesh, and “3..All Nodes”, which would simply perform a node merge for all nodes in the model.

- Updated *Mesh, Connect, Rigid* command by moving the *Independent DOF (Target)* and *Dependent DOF (Source)* options for specifying degrees-of-freedom for the newly created elements to the top of the dialog box. Also, renamed the *Rigid Element Options* section to *Node Pairing Options*, which now contains four options. Finally, the *Preview Target Nodes* option has been moved into the *Target Node Selection* section.




Two of the options which now reside in the *Node Pairing Options* section were simply renamed, *Maximum distance from source to target* (formally *Max Distance*) and *Maximum target(s) per source* (formally *Max Nodes*), which behave the same as they did in previous versions. There are two new options, *Allow target node to be mapped to multiple source nodes*, which allows any target node to be paired with any number of source nodes (if disabled, only pairs with closest source node), and *Minimum target(s) per source*, which when enabled, insures elements will only be created when a source node finds at least the specified number of target nodes using the specified parameters.

- Updated the commands on the *Mesh, Extrude...* menu to use the standard Edge Selection dialog box when Method is set to Along Element Edges, which improves workflow and allows selection of solid element edges.
- Updated the commands on the *Mesh, Sweep...* menu by adding *Added Merge Nodes to Adjacent Elements* and *Limit Sweep to Visible Mesh* to the *Options* section. Both of this options are only available when *Method* is set to *Along Element Edges*. To automatically merge the nodes of the new elements which are created by the sweep to existing adjacent elements, use the *Merge Nodes to Adjacent Elements* option, which is enabled by default. When using *Limit Sweep to Visible Mesh*, which is enabled by default, elements will only be swept along the mesh which is currently visible in the “active” view.

## Elements

- Added “13..Timoshenko” to the *DYNA Options* drop-down of the *Element Formulation* dialog box for Bar and Beam elements.
- Added the ability to view element coordinate systems for *Spring/Damper* elements which reference a property with *Type* set to *CBUSH* by using the *View, Options* command, setting *Category* to *Labels, Entities and Color*, choosing *Element - Coordinate System* from the *Options* list, then enabling *Show Coord Sys*.

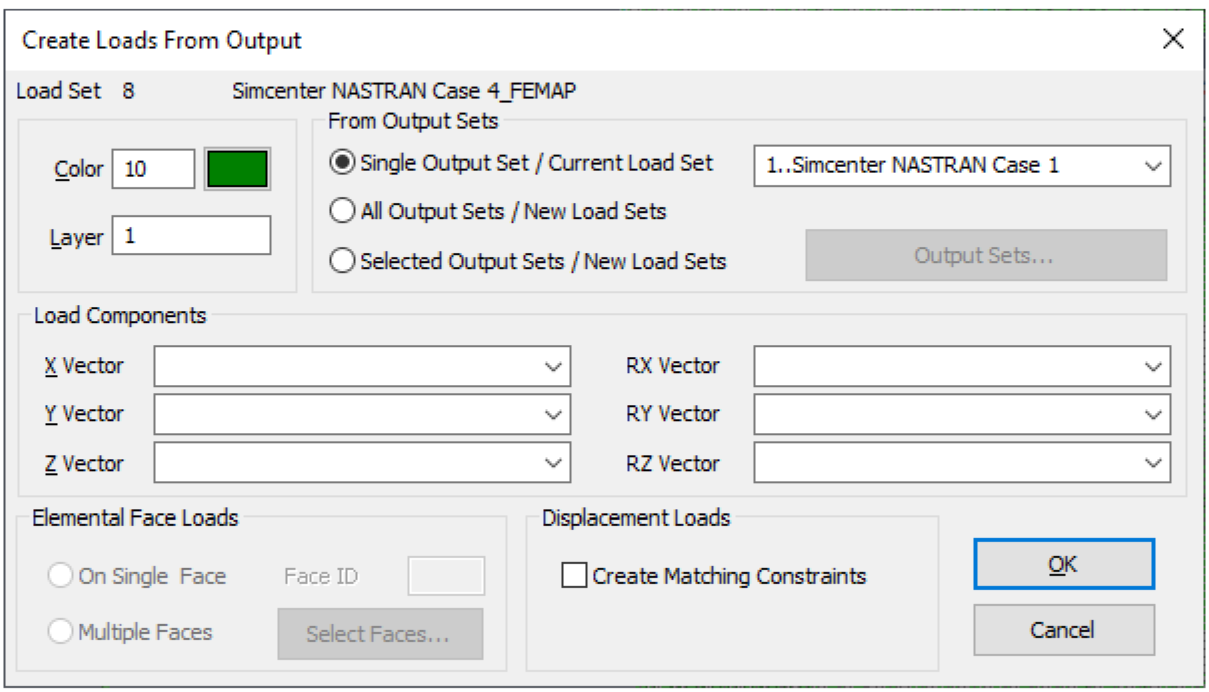
## Loads and Constraints

- Added *Model, Load, Elemental on Face* command, which the same as *Model, Load, Elemental,* except that instead of directly selecting the elements where the loads will be applied, here only the faces of elements are selected via the standard Face Selection dialog box. Once the faces are selected, the command continues, just like the normal *Model, Load, Elemental* command, but only *Pressure*, *Heat Flux*, *Convection*, and *Radiation* loads are available to be applied.
- Updated *Model, Load, From Freebody* command by make additional parameters available for automatic creation of interpolation elements when using the *Multi-Model* option. Also, some of the options from the *Rigid Element Options* section were moved to the new *Node Pairing Options* section, which contains four options.

Two of the options which now reside in the *Node Pairing Options* section were simply renamed, *Maximum distance from source to target* (formally *Max Distance*) and *Maximum target(s) per source* (formally *Max Nodes*), which behave the same as they did in previous versions. There are two new options, *Allow target node to be mapped to multiple source nodes*, which allows any target node to be paired with any number of source nodes (if disabled, only pairs with closest source node), and *Minimum target(s) per source*, which when enabled, insures elements will only be created when a source node finds at least the specified number of target nodes using the specified parameters.

- Updated *Model, Load, From Output* command to allow selection of *All Load Sets* or *Selected Output Sets* to automatically create new load sets using output vector(s) converted into the specified type of load. In addition, added some more robust functionality to specify which element faces to use for *Elemental Face Loads*, along with an option to automatically create matching constraints when creating *Displacement Loads*.

This command converts output data from one or more output vectors into a specified type of load:.




The *Create Loads From Output* dialog box will appear after the type of load to be created has been specified and the user has clicked *OK*. A *Color* and/or *Layer* can be specified for all loads created by this command.

The *From Output Sets* section offers three options:

***Single Output Set/Current Load Set*** - This option creates loads in the active load set using output from a single selected output set.

***All Output Sets/New Load Sets*** - This option creates a new load set for every output set in the model (i.e., if there are 5 output sets in the model, 5 new load sets will be created). Each new load set will contain loads created from the output found in a particular output set..

***Selected Output Sets/New Load Sets*** - This option creates a new load set for every output set selected via the *Output Sets...* button (i.e., if 3output sets are selected, 3new load sets will be created). Each new load set will contain loads created from the output found in a particular output set.

If creating nodal/elemental temperatures, pressures, or heat transfer loads, only a single output vector can be specified in the *Load Components* section, as these types of loads have a single value per node/element/element face.

For the other types of loads, up to six different output vectors can be selected in the *Load Components* section. Data from the six vectors will be converted to the six loading degrees of freedom. If you leave any vectors blank (or zero), no loads will be created in that direction. You must always specify at least one vector.

There are two options available in the *Elemental Face Loads* section when creating loads which are applied to a particular face of an element, such as elemental pressure or many of the elemental heat transfer loads. The *On Single Face* option simply applies the loads to the face specified using the *Face ID* field, for all selected elements. When using the Multiple Faces option, click the Select Faces... button to select element faces using the standard face selection dialog box.

Many solution sequences in Nastran require nodes which have enforced displacements and/or enforced rotations applied to also be constrained in the same degrees of freedom for the loads to be taken into consideration. To automatically create nodal constraints which correspond to enforced displacements and/or enforced rotations being created by the this command, enable the *Create Matching Constraints* option in the *Displacement Load* section. If using either option in the *From Output Sets* section that creates multiple load sets, the same number of corresponding constraint sets will be created.

After everything in the Create Loads from Output dialog box has been specified and the user has clicked *OK*, the standard entity selection dialog box will be displayed. Depending on the type of load selected in the first step of this command, select any number of nodes or elements where loads will be created.

***Note:*** Loads will only be created if output exists for a particular node or element.

## *Simulation Entities - New for 2020.1!*

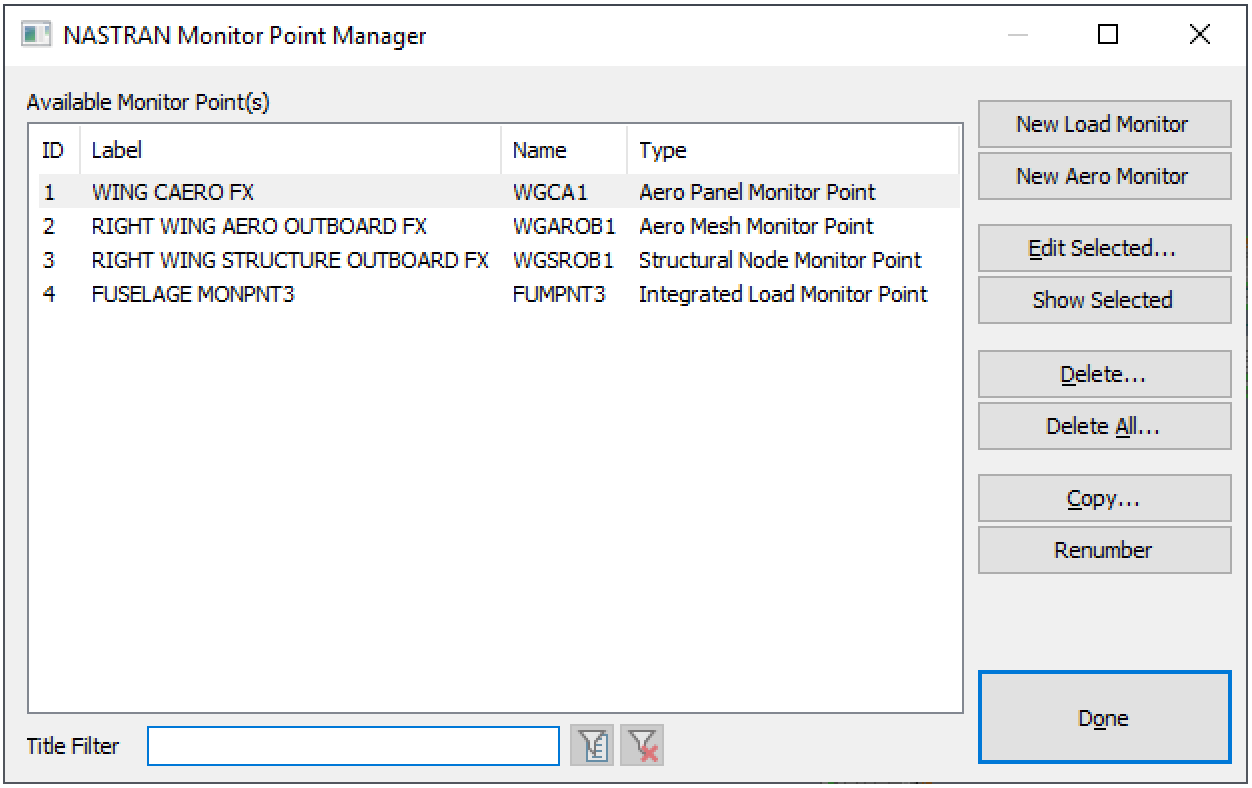
- Added the *Model, Simulation Entities...* menu. The commands on the *Model, Simulation Entities...* menu are used to create, edit, or manage entities, Monitor Points or Direct Matrix Input entities, which can be useful in certain types of analysis performed with Simcenter Nastran or MSC Nastran. These entities are somewhat unique, thus are typically used for more advanced types of analysis and/or by advanced users of Nastran solvers. If these entity types are defined in an existing Nastran input file which has been imported into FEMAP, appropriate entities will be created in FEMAP.
- Added *Model, Simulation Entities, Monitor Points* command, which opens the *NASTRAN Monitor Point Manager* manage monitor point entities in a model.

Monitor Points are only used by Simcenter Nastran and MSC Nastran. There are two distinct types of Monitor Points supported by FEMAP:

***Load Monitor Points*** - create MONPNT3 entries with associated entries. See "NASTRAN Structural Load Monitor Point"

***Aero Monitor Points*** - create MONPNT1 entries with associated entities. See "NASTRAN Aerodynamic Controls Monitor Point"

The *Model, Simulation Entities, Monitor Points* command opens the *NASTRAN Monitor Point Manager*:




This manager is used to create Monitor Points for Simcenter Nastran and MSC Nastran. In FEMAP, there are two types of Monitor Points, “Load Monitor Points”, which create MONPNT3 entries, and “Aero Monitor Points”, which create MONPNT1 entries. “Load Monitor Points” are displayed in the graphics window as a “sigma” surrounded by a square, while “Aero Monitor Points” are displayed as a “sigma” surrounded by a circle.

For Simcenter Nastran, MONPNT3 can only be used in Linear Static and Modal analysis (SOL 101 and SOL 103), while MONPNT1 can only be used for Static Aeroelasticity (SOL 144).

For MSC Nastran, MONPNT3 can be used for all solution sequences except Linear Buckling (SOL 105), Modal Complex Eigenvalue (SOL 110), and some advanced nonlinear and explicit solutions (SOL 600 and SOL 700), while MONPNT1 can be used in Linear Static (SOL 101), Direct and Modal Frequency Response (SOL 108 and SOL 111), Direct and Modal Transient Response (SOL 109 and SOL 112), Static Aeroelasticity (SOL 144), Aeroelastic Response (SOL 146), and Design Optimization (SOL 200).

***Note:*** Some of the solution sequences mentioned above for MSC Nastran, SOL 146, SOL 200 (for Aeroelasticity), SOL 600, and SOL 700, are not directly supported by FEMAP.

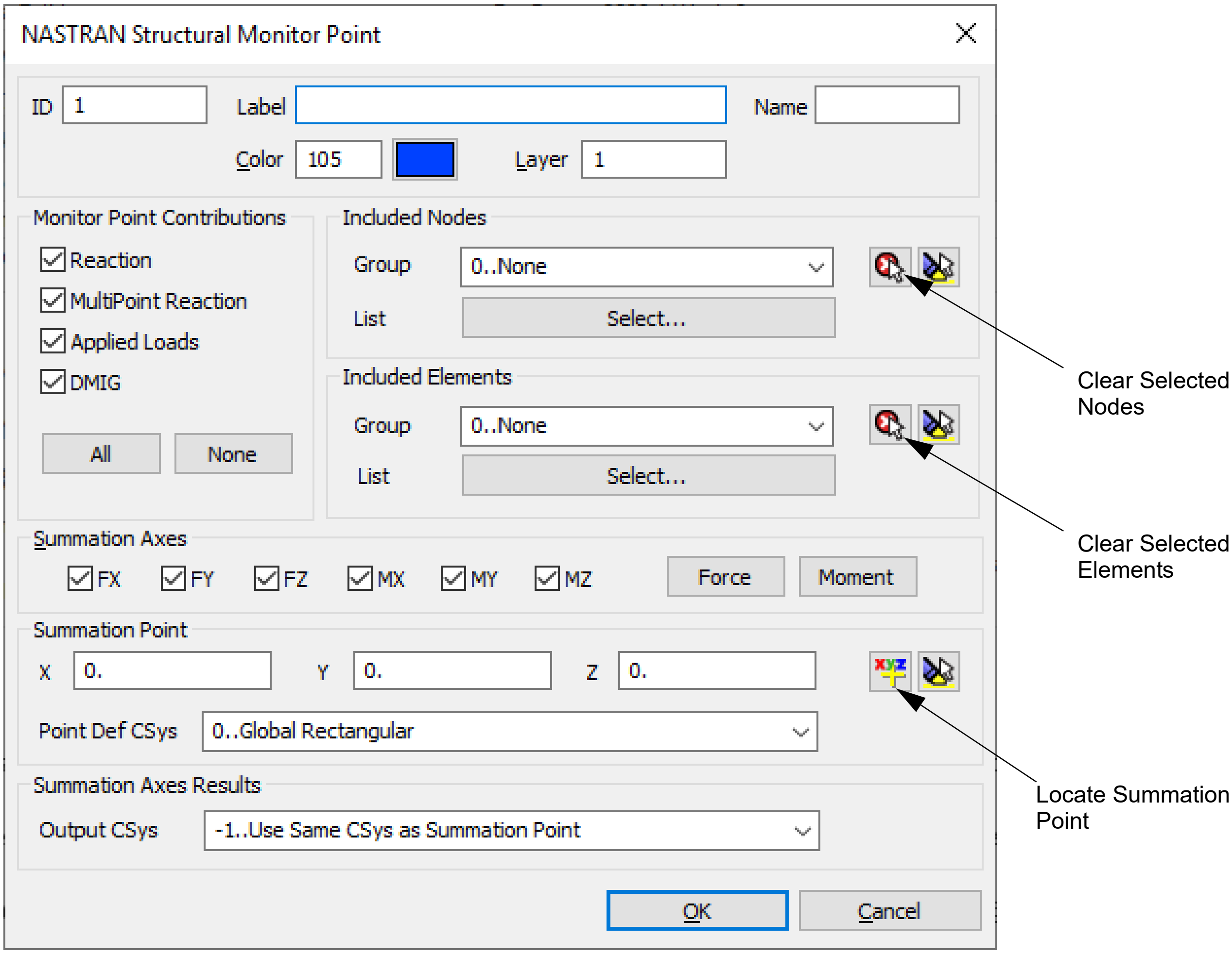
This manager can be used to create a *New Load Monitor* or *New Areo Monitor*; *Edit, Delete*, *Copy*, or *Renumber* the monitor point currently highlighted in the *Available Manufacturing constraint(s)* list; highlight entities in the graphics window which are referenced by the monitor point(s) currently highlighted in the list (*Show Selected*); or *Delete All* the monitor points currently in the model.

Clicking the *New Load Monitor* button opens the *NASTRAN Structural Load Monitor Point* dialog box, which is used to create MONPNT3 entries. See "NASTRAN Structural Load Monitor Point" for more information. Clicking the *Load Aero Monitor* button opens the *NASTRAN Aerodynamic Control Monitor Point* dialog box, which is used to create MONPNT1 entries. See "NASTRAN Aerodynamic Controls Monitor Point" for more information.

Once monitor points have been created, one additional step is required. To be written to the Nastran input file, a monitor point must be selected via the *Select Monitor Point(s)* dialog box, which can be accessed in the *Options* section of the Master case or any subcase of an appropriate analysis set. For more information about appropriate analysis types, as they differ between Simcenter Nastran and MSC Nastran, please see above.

### *NASTRAN Structural Load Monitor Point*

This dialog box is used to create MONPNT3 entries for Simcenter Nastran and MSC Nastran:




An *ID*, *Title* (up to 56 character), *Color*, and/or *Layer* can be specified for the monitor point, otherwise, the next available ID will be used, no title will be assigned, the default color for monitor points used, and the entity placed on the layer currently active in the model.

Unlike many other entity types in FEMAP, a unique *Name* (up to 8 characters) MUST be specified for each monitor point. The specified Name NAME field on the MONPNT3 entry.

***Note:*** If a *Title* is specified, it is limited to 56 characters. This is because any specified *Title* will be written to the MONPNT3 entry in the LABEL field, which is limited to 56 characters. Unlike *Name*, the *Title* does not have to be unique, but this is recommended.

The *Monitor Point Contributions* section is used to choose which contributions, *Reaction* (SPC Forces), *MultiPoint Reaction* (MPC Forces), *Applied Loads*, and/or DMIG (DMIG entries at the monitor point), will be included in the summation calculation at the monitor point. By default, all contributions are enabled, which writes nothing to the XFLAG field of the MONPNT3 entry. When a contribution is disabled, a specific character will be written to the XFLAG field (S for *Reaction*, M for *MultiPoint Reaction*, A for *Applied Loads*, D for *DMIG*) and any combination of S, M, A, and/or D is allowed. The *All* and *None* buttons in this section can be used to quickly toggle all contributions on or off, respectively.

The *Included Nodes* section is used to specify which nodes will be included in the monitor point. This can be done by selecting an existing group from the *Group* drop-down or using the Select button to the right of *List* to select nodes using the standard entity selection dialog box. If selecting nodes with the *Select* button, the number of "Nodes" in the List will replace "Select" on the button. At any time, the "Clear Selected Nodes" icon button can be used to remove all nodes currently in the *List* of selected nodes. In addition, the "Preview Included Nodes" icon button can be used to highlight the included nodes in the graphics window, using the current settings for the *Win-*

*dow, Show Entities* command. The *Included Elements* section is very similar to the *Included Nodes* section, but elements are selected to include instead of nodes.

Regardless of how the nodes have been included for the monitor point entry in FEMAP, a GROUP entry with "GRID" or "specified for TYPEi (Simcenter Nastran) or a SET1 entry (MSC Nastran) and the included node IDs will be written to the Nastran input file. The ID of this GROUP or SET1 entry is then referenced by MONPNT3 using the GRIDGRP field (Simcenter Nastran) or GRIDSET field (MSC Nastran). For elements, a GROUP entry with "ELEM" specified for TYPEi (Simcenter Nastran) or a SET1 entry (MSC Nastran) and the included element IDs will be written to the Nastran input file. The ID of this GROUP or SET1 entry is then referenced by MONPNT3 using the ELEMGRP field (Simcenter Nastran) or ELEMSET field (MSC Nastran). It is not necessary to specify which elements to include, but if no elements are specified, the elements attached to the included nodes are excluded from the summation.

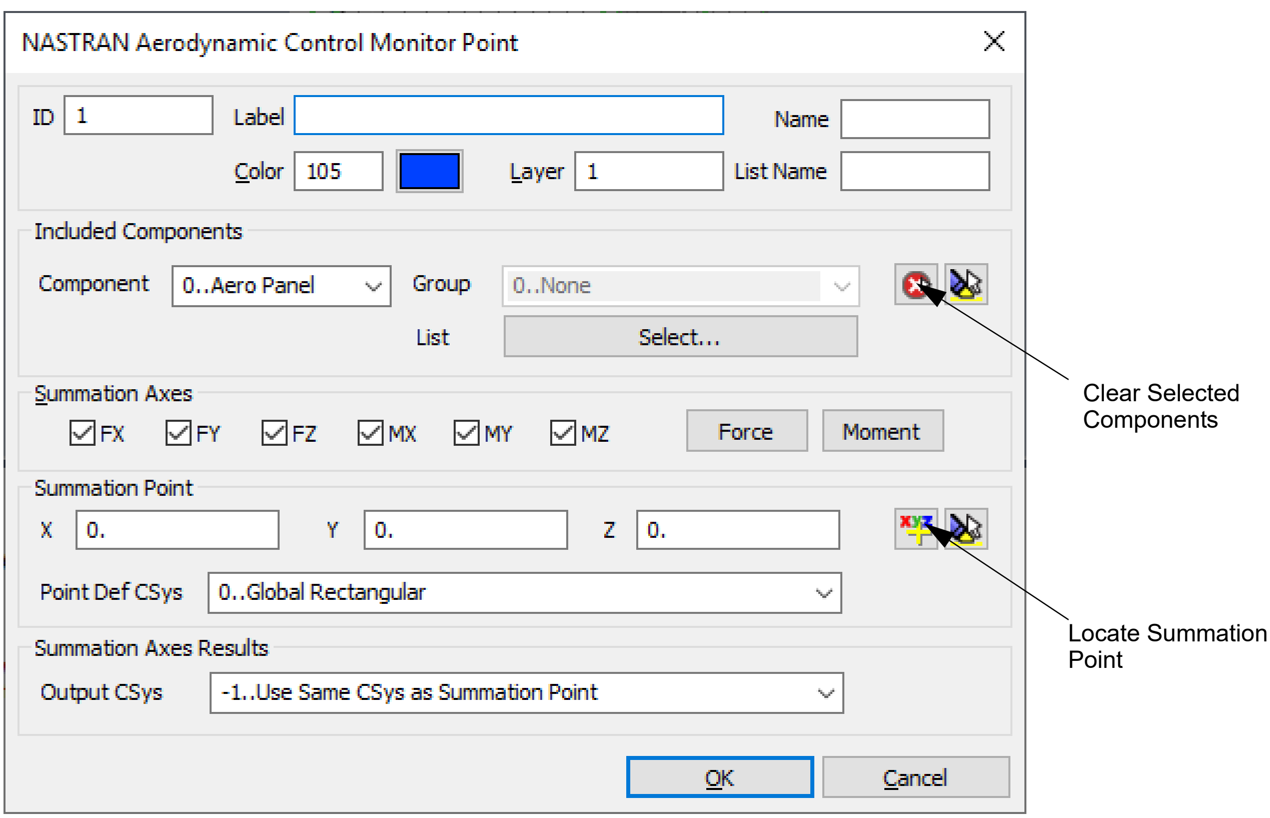
The *Summation Axes* section contains the component axes about which to sum. Any combination of the check boxes in this section being enabled or disabled is allowed, but at least one must be enabled. The *Force* button can be used to quickly toggle the *FX*, *FY*, and *FZ* check boxes on/off, while the *Moment* button can be used to the same for the *MX*, *MY*, and *MZ* check boxes. Based on which axes are enabled, a combination of integers 1 through 6 with no embedded blank spaces will be written to the AXES field of the MONPNT3 entry.

The *Summation Point* section is used to specify the coordinates about which the forces are to be summed. These coordinates are defined in relation to the coordinate system selected via the *Point Def CSys* drop-down. The "Locate Summation Point" icon button is used to select a location from the graphics window, while the "Preview Summation Point Location" icon button is used to show the currently specified location in the graphics window. With regards to the MONPNT3 entry, the ID of the coordinate specified for *Point Def CSys* will be written to the CP field, while *X*, *Y*, and *Z* will be written to X1, X2, and X3 (Simcenter) or X, Y, and Z (MSC) fields, respectively.

Finally, the *Summation Axes Results* section is used to specify a coordinate system for the results to be output via the *Output CSys* drop-down. By default, this is typically set to "-1..Use Same CSys as Summation Point", but any existing coordinate system can be selected. When using "-1..Use Same CSys as Summation Point", the ID of the coordinate system specified for *Point Def CSys* to the CD field on the MONPNT3, otherwise, the ID specified for *Output CSys* is written to the CD field.

### NASTRAN Aerodynamic Controls Monitor Point

This dialog box is used to create MONPNT1 entries for Simcenter Nastran and MSC Nastran:



An *ID*, *Title* (up to 56 character), *Color*, and/or *Layer* can be specified for the monitor point, otherwise, the next available ID will be used, no title will be assigned, the default color for monitor points used, and the entity placed on the layer currently active in the model.

Unlike many other entity types in FEMAP, a unique *Name* (up to 8 characters) MUST be specified for each monitor point. The specified Name NAME field on the MONPNT1 entry. Also, the COMP field of the MONPNT1 references an AECOMP entry by a “List Name”. While the “List Name” can be the same as NAME (*Name* in FEMAP), it does not have to be, thus a different “List Name” can be specified using *List Name* (up to 8 characters).

***Note:*** If a *Title* is specified, it is limited to 56 characters. This is because any specified *Title* will be written to the MONPNT1 entry in the LABEL field, which is limited to 56 characters. Unlike *Name*, the *Title* does not have to be unique, but this is recommended, as it is used in printed output.

The *Included Components* section is used to specify which Aero Panels, Aero Mesh entities, or nodes will be included in the monitor point. When *Component* is set to:

**“0..Aero Panel”** - Only the *Select* button next to *List* will be available to select Aero Panel entities using the standard selection dialog box. After selection, the number of “Panels/Bodies” in the List will replace “Select” on the button. An AECOMP entry with a NAME field that matches with *Name* or *List Name* on the monitor point in FEMAP, LISTYPE set to CAERO, and the IDs of the selected aero panels will be written to the Nastran input file. This AECOMP entry is then referenced by the MONPNT1 entry via the COMP field.

**“1..Aero Mesh”** - Only the *Select* button next to *List* will be available to select Aero Mesh entities using the standard selection dialog box and, once selected, the number of selected “Aero Elements” will replace “Select...” on the button. An AELIST entry containing the aero element IDs is written to the Nastran input file, along with a corresponding AECOMP entry with a NAME field that matches *Name* or *List Name* on the monitor point in FEMAP, LISTTYPE set to AELIST, and the ID of a single AELIST entry. This AECOMP entry is then referenced by the MONPNT1 entry via the COMP field.

**“2..Nodes”** - Nodes can be included by either selecting an existing group from the *Group* drop-down or using the *Select* button to the right of *List* to select nodes using the standard entity selection dialog box. If selecting nodes with the *Select* button, the number of nodes in the List will replace “Select” on the button. Regardless of how the nodes have been included for the monitor point entry in FEMAP, a SET1 entry containing node IDs is written to the Nastran input file, along with a corresponding AECOMP entry with a NAME field that matches *Name* or *List Name* on the monitor point in FEMAP, LISTTYPE set to SET1, and the ID of a single SET1 entry. This AECOMP entry is then referenced by the MONPNT1 entry via the COMP field.

For all types, the “Clear Selected Components” icon button can be used to remove all entities currently in the *List* of selected entities. In addition, the “Preview Included Components” icon button can be used to highlight the included entities in the graphics window, using the current settings for the *Window, Show Entities* command.

The *Summation Axes* section contains the component axes about which to sum. Any combination of the check boxes in this section being enabled or disabled is allowed, but at least one must be enabled. The *Force* button can be used to quickly toggle the *FX*, *FY*, and *FZ* check boxes on/off, while the *Moment* button can be used to the same for the *MX*, *MY*, and *MZ* check boxes. Based on which axes are enabled, a combination of integers 1 through 6 with no embedded blank spaces will be written to the AXES field of the MONPNT1 entry.

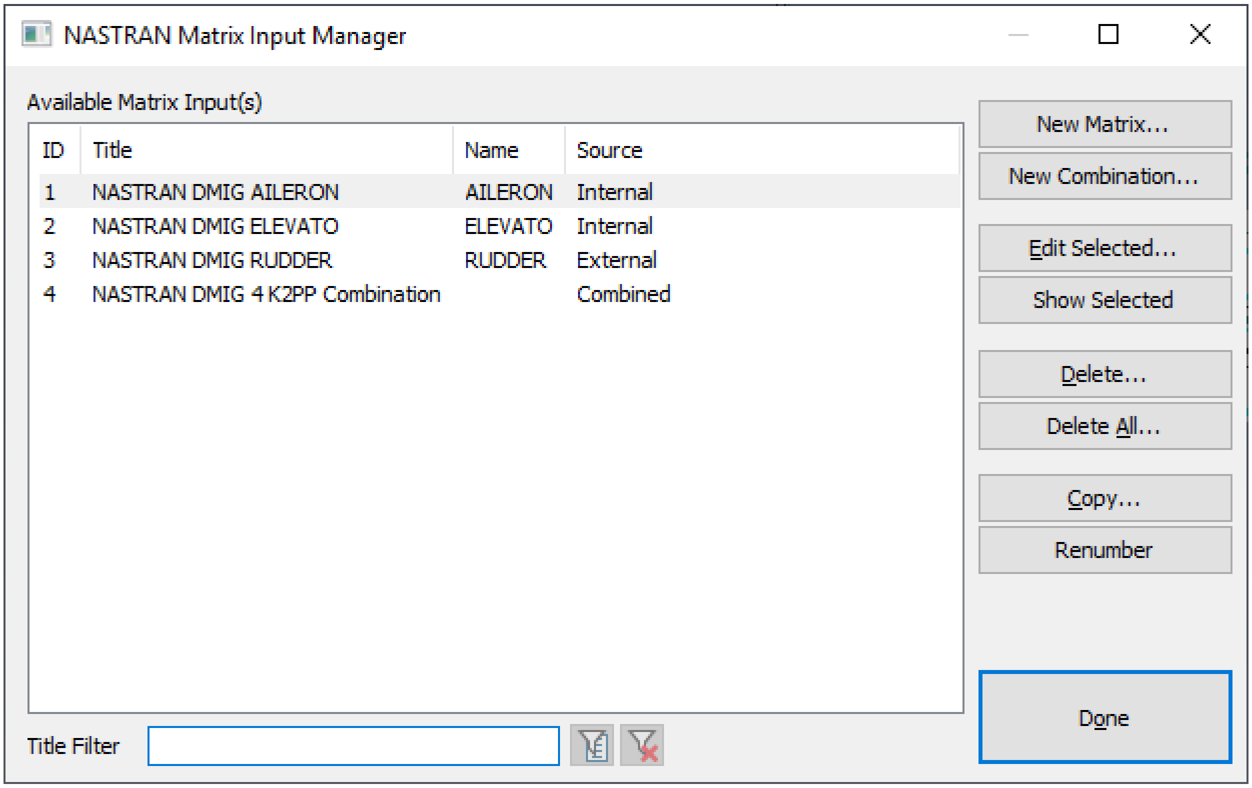
The *Summation Point* section is used to specify the coordinates about which the forces are to be summed. These coordinates are defined in relation to the coordinate system selected via the *Point Def CSys* drop-down. The “Locate Summation Point” icon button can be used to select a location from the graphics window, while the “Preview Summation Point Location” icon button can be used to show the currently specified location in the graphics window. With regards to the MONPNT1 entry, the ID of the coordinate specified for *Point Def CSys* will be written to the CID field (Simcenter Nastran) or CP field (MSC Nastran), while *X*, *Y*, and *Z* will be written to the X, Y, and Z fields, respectively.

Finally, the *Summation Axes Results* section is used to specify a coordinate system for the results to be output via the *Output CSys* drop-down. By default, this is typically set to “-1..Use Same CSys as Summation Point”, but any existing coordinate system can be selected. When using “-1..Use Same CSys as Summation Point”, the ID of the coordinate system specified for *Point Def CSys* to the CD field on the MONPNT1, otherwise, the ID specified for *Output CSys* is written to the CD field.

**Note:** The *Output CSys* entry in the *Summation Axes Results* section will only be written out for MSC Nastran, as this option is not currently available in Simcenter Nastran.

- Added *Model, Simulation Entities, Direct Matrix Input* command, which opens the *NASTRAN Matrix Input Manager*, which is used to manage direct matrix input entities in a model.

This command Opens the *NASTRAN Matrix Input Manager*:




There are two types, single Direct Matrix Input entities and combinations of multiple Direct Matrix inputs.

This manager is used to create Direct Matrix Input entities (DMIGs) for Nastran solvers. In FEMAP, there are two types of direct matrix inputs, “Single Matrix”, which writes a single DMIG entry and corresponding case control entry, and “Matrix Combination”, which writes all of the DMIG entries referenced by the combination, along with a corresponding case control entry.

This manager can be used to create a *New Matrix* or *New Combination*; *Edit, Delete*, *Copy*, or *Renumber* the direct matrix input(s) currently highlighted in the *Available Matrix Input(s)* list; highlight entities in the graphics window which are referenced by the direct matrix input(s) currently highlighted in the list (*Show Selected*); or *Delete All* the direct matrix inputs currently in the model.

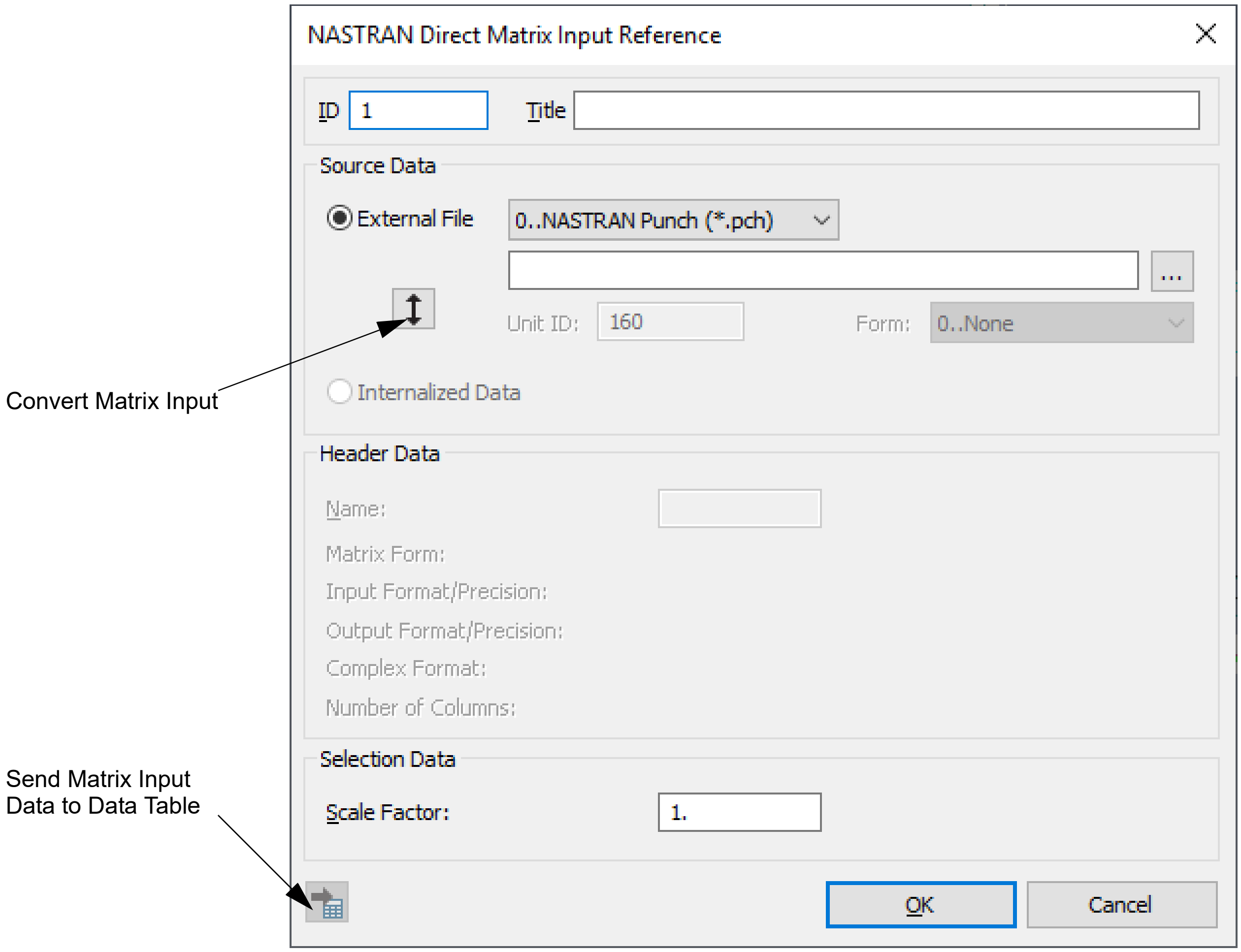
Clicking the *New Matrix* button opens the *NASTRAN Direct Matrix Input Reference* dialog box, which is used to create either an actual DMIG entry in the Nastran input file (*Internalized Data*) or an INCLUDE statement to the Nastran input file which references an external file containing a DMIG entry (*External File*). See "NASTRAN Direct Matrix Input Reference" for more information.

Clicking the *New Combination* button opens the *Referenced Matrix Inputs for NASTRAN Matrix Selection* dialog box, which is used to create a combination of direct matrix inputs, which can include scale factors for each referenced direct matrix input entity. Each combination writes any DMIG entry and/or INCLUDE statement required for the combination, provided they are not being written out to the Nastran input file as a “Single Matrix” (i.e., will only be written out once). See "Referenced Matrix Inputs for NASTRAN Matrix Selection" for more information.

Once the direct matrix input entities have been created, one additional step is required. To be written to the Nastran input file, a direct matrix input, “Single Matrix” or “Matrix Combination”, must be selected via the *NASTRAN Matrix Input Selection* dialog box, which can be accessed for the Master case or any subcase of an appropriate analysis set. The type of case control entry written for a direct matrix input entity depends on which drop-down references the entity in the *NASTRAN Matrix Input Selection* dialog box.

### *NASTRAN Direct Matrix Input Reference*

This dialog box is used to create DMIG entries for Nastran solvers:




An *ID* and *Title* can be specified for the Direct Matrix Input, otherwise, the next available ID will be used and no title will be assigned.

The *Source Data* section is used to specify if the DMIG information exists in *External File* or is saved to the FEMAP database as *Internalized Data*. In the large majority of usage cases, the DMIG information has been automatically generated by a Nastran solver, thus it already exists in a file. Because of this, there is no ability to manually create a “Matrix of Values” to use as a DMIG via the FEMAP User Interface. Instead, FEMAP offers the ability to select an external file, then use the “Convert Matrix Input” icon button to internalize the data or use internalized data to create to create an external file.

When *Source Data* is set to External File, the user must first choose the type of file from the drop-down, either “0..NASTRAN Punch (*.pch)” or “1..NASTRAN OP2 (*.op2)”, then use the “...” icon button to open a dialog box to select the file. When using “1.NASTRAN OP2 (*.op2)”, the *Unit ID* and *Form* options become available, which are used to optionally specify an ID of a FORTRAN Unit and/or form of ASSIGN statements, respectively.

After a file has been selected and/or internalized, the *Header Data* section will be populated with:

***Name*** - Name of the Matrix (Up to 8 alphanumeric characters, the first of which is alphabetic), which is written to/read from the NAME field of the DMIG entry.

***Matrix Form*** - Form of Matrix Input. Options are “Square” (Value on DMIG = 1), “Rectangular 2” (2), “Rectangular 9” (9), or Symmetric” (6), which are read from/written to IFO field of the DMIG entry.

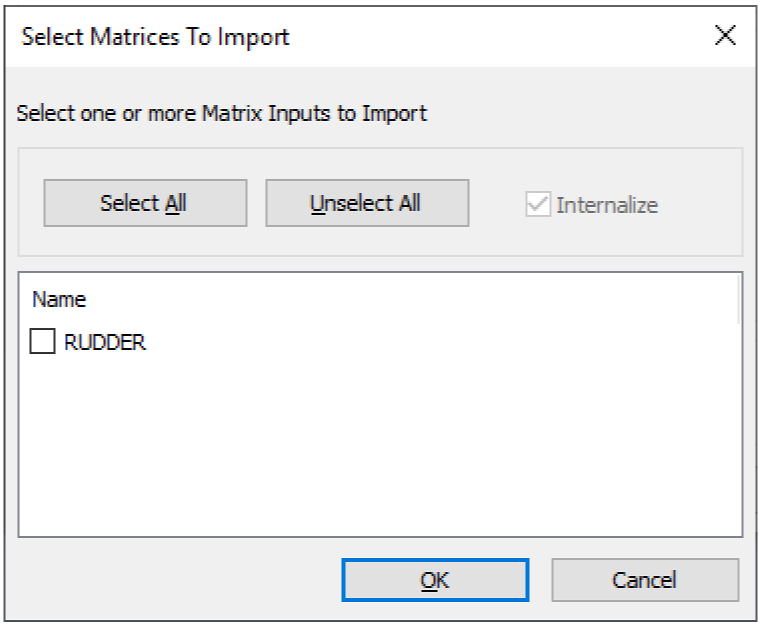
***Input Format/Precision*** - Type of matrix being import. Options are “Real, Single Precision” (Value on DMIG = 1), “Real, Double Precision” (2), “Complex, Single Precision” (3), or “Complex, Double Precision” (4), which are read from/written to the TIN field on the DMIG entry.

***Output Format/Precision*** - Type of matrix that will be created. Options are “Machine Precision” (Value on DMIG = 0), “Real, Single Precision” (1), “Real, Double Precision” (2), “Complex, Single Precision” (3), or “Complex, Double Precision” (4), which are read from/written to the TOUT field on the DMIG entry.

***Complex Format*** - Input format of Ai, Bi for a complex matrix. Options are “Real, Imaginary” (Value on DMIG = 0) or “Amplitude, Phase” (1), which are read from/written to the POLAR field of the DMIG entry.

***Number of Columns*** - Number of columns in a rectangular matrix, which is only used when the *Matrix Form* is set to “Rectangular 9” (IFO = 9). Value is read from/written to the NCOL field of the DMIG entry.

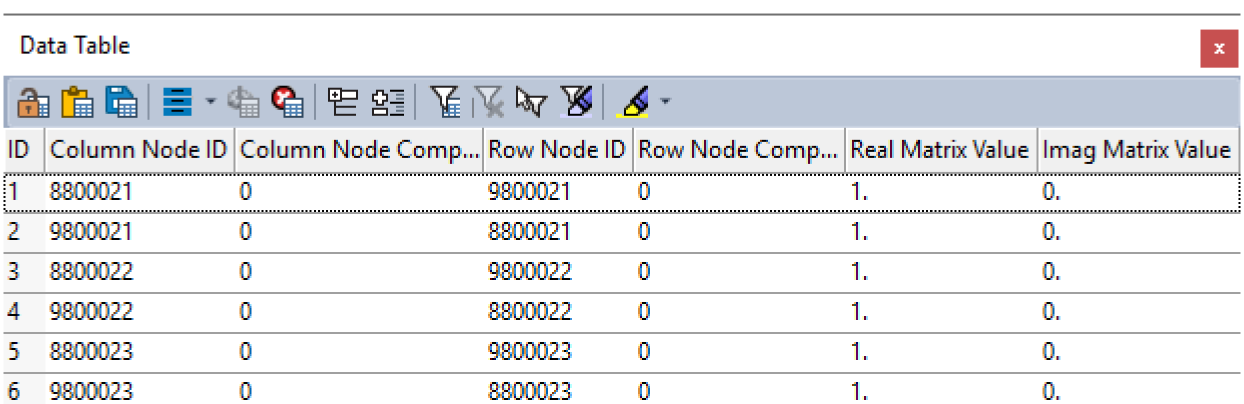
When internalizing matrices, the *Select Matrices to Import* dialog box will appear:




Simply check the box next to the name of each matrix to be internalized, then click *OK*. Use *Select All* to quickly select all of the DMIG names in the external file or use *Unselect All* to clear all of the check boxes. A separate direct matrix input entity will be created for each item selected in the dialog box.

The *Selection Data* section contains a single option, *Scale Factor*, which can be used to scale the direct matrix input entity without the need to create a “Matrix Combination”, as that could be somewhat cumbersome. In addition, if a direct matrix input entity is highlighted when making a “Matrix Combination”, the *Scale Factor* field in the *Referenced Matrix Inputs for NASTRAN Matrix Selection* dialog box will be populated with this value.

The “Send Matrix Input Data to Data Table” icon button will only be available when *Source Data* is set to *Internalized Data*. Clicking this button will send the direct matrix input data to the *Data Table* in a format similar to this:

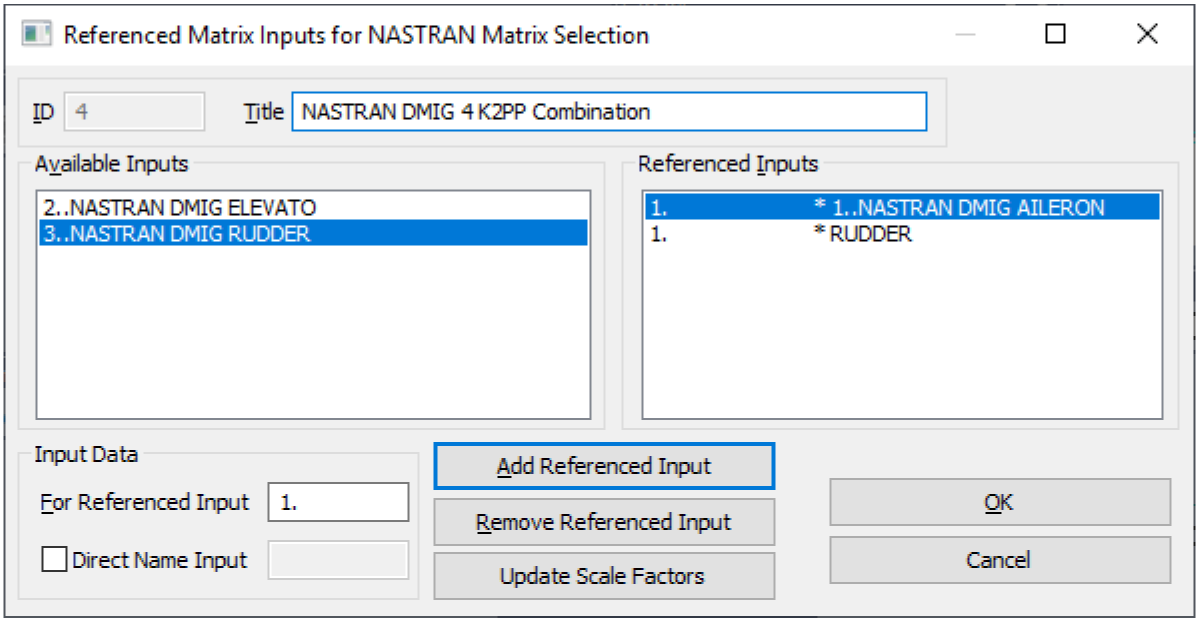


Data Table

| ID | Column Node ID | Column Node Comp... | Row Node ID | Row Node Comp... | Real Matrix Value | Imag Matrix Value |
|---|---|---|---|---|---|---|
| 1 | 8800021 | 0 | 9800021 | 0 | 1. | 0. |
| 2 | 9800021 | 0 | 8800021 | 0 | 1. | 0. |
| 3 | 8800022 | 0 | 9800022 | 0 | 1. | 0. |
| 4 | 9800022 | 0 | 8800022 | 0 | 1. | 0. |
| 5 | 8800023 | 0 | 9800023 | 0 | 1. | 0. |
| 6 | 9800023 | 0 | 8800023 | 0 | 1. | 0. |

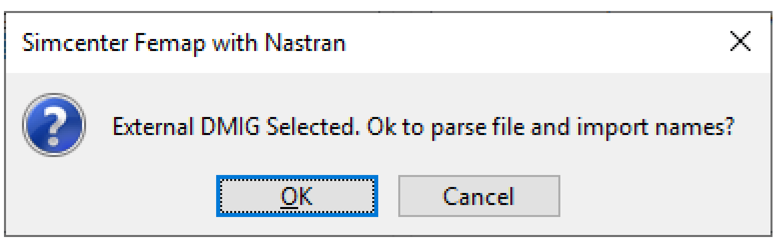
### *Referenced Matrix Inputs for NASTRAN Matrix Selection*

This dialog box is used to create Matrix Combination entries:




Highlight any number of direct matrix inputs from the list of *Available Inputs*. Click *Add Referenced Input* to have them placed in the *Referenced Inputs* list. By default, each item placed into the *Referenced Inputs* list will be included with a *Scale Factor* of “1.0”, unless another value has been specified for *Scale Factor* in the *Selection Data* section of the *NASTRAN Direct Matrix Reference* dialog box for the direct matrix input. If desired, the *For Referenced Input* “Scale Factor” can be changed before pressing the *Add Referenced Input* button and all highlighted load sets will be placed in the *Referenced Inputs* list using that “Scale Factor”. These scale factors will be written to the case control section, along with the names of the referenced inputs. *Update Scale Factors* will update the scale factors of all inputs currently highlighted in the *Referenced Inputs* list, while the *Remove Referenced Input* button is used to remove highlighted load sets from the *Referenced Inputs* list.

Additional user input is required when attempting to add a direct matrix input entity which has *Data Source* set to *External File*. After pressing *Add Referenced Input*, a question will appear:




After clicking *OK*, the *Select Matrices to Import* dialog box, which is described above, will appear with one additional option, *Internalize*. When *Internalized* is disabled, which is the default, only the names will be imported. When *Internalize* is enabled, a new direct matrix input entity will be created based on the selected matrix name(s).

Alternatively, a DMIG entry located in an external file can be added to a matrix combination by enabling the *Direct Name Input* option, entering the DMIG entry’s NAME, then clicking the *Add Referenced Input* button.

Any direct matrix input entity which has *Data Source* set to *External File* will appear the same in the *Referenced Inputs* list, regardless of the method used to add it to the matrix combination.

## *Connections (Regions, Properties, and Connectors)*

- Updated **Adaptively Modify Penalty Factor* option on *Multistep Structural (401)* tab, which is accessed by clicking *More Options...* button. Previously, it was a check box, but now it is a drop-down with three options.

***Adaptively Modify Penalty Factor** - Creates PENADAPT field in BCTPARM entry. Option to adaptively vary the contact stiffness between iterations.

**"0..Do Not Modify"** - No special field will be written to the BCTPARM and the penalty factor is not adaptively modified by the program, which is the default behavior.

**"1..Adaptively Modify"** - Penalty factor is adaptively modified by the program. Writes a value of 1.

**"2..Adaptively Modify in Large Range"** - Penalty factor is adaptively modified in a range larger than when this option is set to "1..Adaptively Modify". Generally, "1..Adaptively Modify" is a good choice for most problems. However, for some cases you may need a wider range of penalty factors. For these cases, you should consider using this option. Writes a value of 2.

- Added *Normal Vel Coeff.* drop-down, *Tangent Vel Coeff.* drop-down, *Normal Reg. Type* drop-down and *Value*, *Frict vs Time* drop-down, and *Frict vs Temp* drop-down to the *Multistep Kinematic (402)* tab.

**Normal Vel Coeff.** - Choose an existing function from the drop-down, which describes the normal velocity coefficient vs the normal penetration distance. This will write the DPARA1, along with the ID of the selected function, to the BCTPAR2. For the function, the units for the X-Axis are Length and units for the Y-Axis, Pressure/Speed (Force*Time/Length$^3$). For a given normal penetration distance, the value of the table is then multiplied by the normal velocity to produce a viscous pressure.

**Tangent Vel Coeff.** - Choose an existing function from the drop-down, which describes the tangential velocity coefficient vs the normal penetration distance. This will write the DPARA2, along with the ID of the selected function, to the BCTPAR2. For the function, the units for the X-Axis are Length and units for the Y-Axis, Pressure/Speed (Force*Time/Length$^3$).For a given normal penetration distance, the value of the table is then multiplied by the tangential sliding velocity to produce a viscous pressure.

**Normal Reg. Type** and **Value** - Creates PRCS field in BCTPAR2 entry. Specifies normal regularization type. Default value is -3.

**"-3..Automatic"** - Automatic choice: if the characteristic stiffnesses of both contact supports are very different, the characteristic stiffness of each support is used, otherwise the characteristic stiffness of the whole structure is used.

**"-2..Use Contact Support Stiffness"** - use the characteristic stiffness of each contact support instead of the stiffness of the whole structure.

**"-1..No Regularization"** - use no regularization.

**"0..Specify Regularization Value"** - define the regularization value to use by entering an integer value above 0 in the *Value* field.

**Frict vs Time** - Choose an existing function from the drop-down, which describes the relation between the friction coefficient vs time. This will write the CFNF, along with the ID of the selected function, to the BCTPAR2. For the function, the units for the X-Axis, Time and units for the Y-Axis, Unitless. Used by Friction Model 1.

**Frict vs Temp** - Choose an existing function from the drop-down, which describes the relation between the friction coefficient vs temperature. This will write the CFTE, along with the ID of the selected function, to the BCTPAR2. For the function, the units for the X-Axis are Temperature and units for the Y-Axis, Unitless. Used by Friction Model 1.

- Added *Target KEYOPTs* section and all options in the section to the *ANSYS* tab. This section contains options which are only for target segments (TARGE169 and TARGA170). Of particular interest is *Pilot Constraint (4)*, which can be used to specify constrained degrees of freedom for target segments which reference a particular connection property.
- Added *Thermal* section to *MSC Nastran* tab, which is accessed by clicking the *Advanced Options...* button. Options in the *Thermal* section are written to the "HHHB" line in the BCTABLE entry.
- Updated the title of the tab in the *Define Connection Property* dialog box used to specify connection property values for *Autodesk Nastran* (formally *NEi Nastran*).

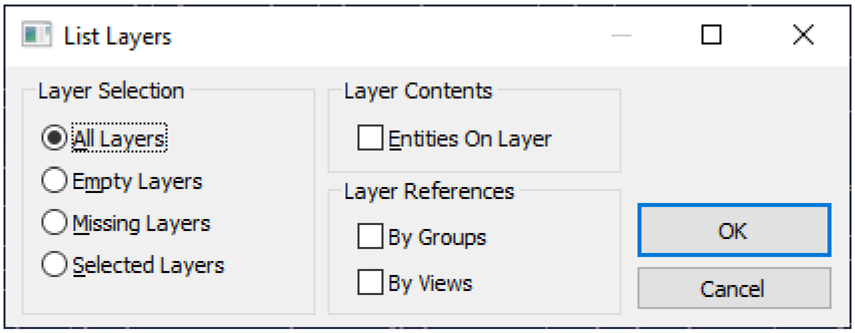
## Aeroelasticity

- Added the ability to preview the *Aero Mesh* to the for *Control Surface 1* and *Control Surface 2* to the *Create Aero Control Surface* dialog box.
- Added support for MONPNT1 for Simcenter Nastran and MSC Nastran. See "Simulation Entities - New for 2020.1!" for more information.

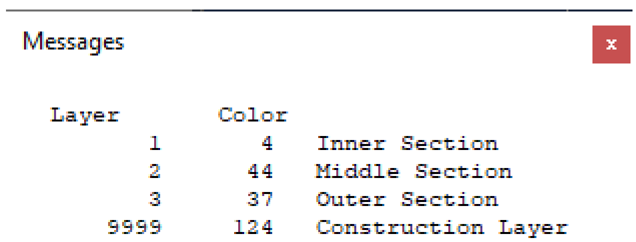
## Listing

- Updated *List, Tools, Layers* to optionally list entities on layers, groups with reference layers, and/or layers which are a visible layer by views. In addition, can now also list “missing layers” and “empty layers”.

This command produces a report which describes all layers defined in a model. Opens the *List Layers* dialog box:




By default, the *Layer Selection* option is set to *All Layers*, which when used will simply create a listing of all layers which exist in your model, with each layer’s ID and Color:




To include the entities assigned to each layer, enable the *Entities on Layer* option in the *Layer Contents* section.

To list any group(s) which reference a layer, if any, enable *By Groups* in the *Layer References* section.

To list any view(s) which the layer is currently visible, if any, enable *By Views* in the *Layer References* section.

***Note:*** If the overall layer display option for a view is set to *View All Layers*, then that view will not listed as a view which has a layer visible. This is only used when the overall layer display option is set to *View Visible Layers Only* and a layer is “Visible”.

The other options in the *Layer Selection* section do the same as *All Layers*, but can be used to list:

***Empty Layers*** - layers which have no entities assigned, but none of other listing options are available.

***Missing Layers*** - layers which have at least one entity assigned, but the layer does not exist in the model. The *Entities on Layer* and *By View* options are also available when using this option.

***Selected Layers*** - layers selected by the user via the Select *Layers(s) to List* dialog box. All other listing options are available when using this option.

Here is a example of using the *Selected Layers* option with *Entities on Layer*, *By Group*, and *By View*, all enabled:

```
Messages
List Layers
Layer 2 - Middle Section         Color: 44
  Node(s)
           77               ,        90              ,        95
          174               ,       177 thru      178,       181 thru       182
          185 thru       186,       190              ,       238 thru       240
          243 thru       245,       330              ,       334 thru       335
          339               ,       370 thru      371,       374
  Element(s)
            4 thru         6,        13 thru       15,        25 thru        30
           45 thru        52,        63 thru       64,        69 thru        70
           85 thru        96,       121 thru      132,       153 thru       160
          175 thru       180,       193 thru      198,       208 thru       210
          217 thru       219,       226 thru      228,       235 thru       237
          244 thru       246,       253 thru      255
  Referenced By Group(s)
           11
  Referenced By View(s)
            1

==========================================================================================
```

## Groups and Layers

- Updated *List, Tools, Layers* to optionally list entities on layers, groups with reference layers, and/or layers which are a visible layer by views. In addition, can now also list “missing layers” and “empty layers”. See Listing section for more information.
- Added ability to use entity-specific commands on the *Group* menu, for any entity type which exists in the model, even if no groups currently exist in the model. When a *Group* menu command is used in this situation, the user will simply be prompted to create a new group.

## Views

- Implemented “Unified Label Architecture”, which is used for all graphics, regardless if entities are being drawn with Performance Graphics, “Legacy OpenGL Graphics”, or a mix of both. This implementation includes enhanced control of digits for real number values being displayed in the graphics window.

To set global settings for digit control, use the *File, Preferences* command, choose the *Graphics* tab, then use then specify options using the *Locale* and *Options...* buttons in the *Digits Control* section. See Preferences section for more information.

The following options in *View, Options* contain a *Digits* button which can be used to override the options set in the *Digit Options* dialog box accessed via the *File, Preferences*:

***Labels, Entities and Color* Category** - *Load Vectors* and *Constraint*

***Tools and View Style* Category** - None

***PostProcessing* Category** - *Deformed Model*, *Contour/Criteria Style*, *Contour/Criteria Legend*, *Contour Arrow Options*, and *Freebody Vectors*

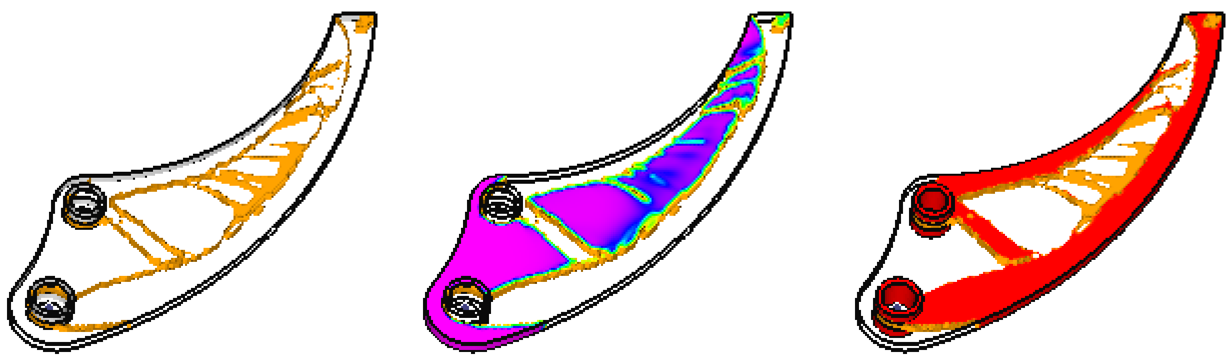
The default setting for digit control for all of the above view options is “Use Preferences”. All of the other available settings are the same as the options available via the *Options...* button in the preferences, so again, see Preferences section for more information on the available settings.

- Added the ability to view element coordinate systems for *Spring/Damper* elements which reference a property with *Type* set to *CBUSH* by using the *View, Options* command, setting *Category* to *Labels, Entities and Color*, choosing *Element - Coordinate System* from the *Options* list, then enabling *Show Coord Sys*.

## Output and Post-Processing

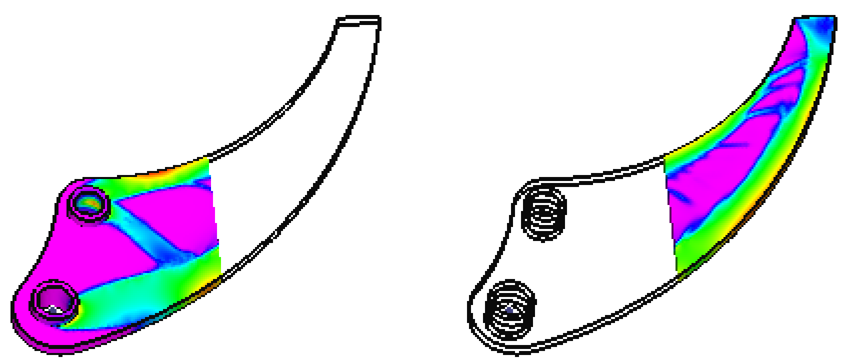
- Output Vector ID Ranges have been modified. See VERY IMPORTANT - Output Vector ID Ranges Have Changed! section above.
- Added the ability when plotting an IsoSurface which allows a single IsoSurface to be used as a “cap”, which will contour all elements and portions of elements which have a value below the specified value (“2..Negative Capped Isosurface”) or above the specified value (“3..Positive Capped Isosurface”). This option can be controlled by the *IsoSurface* option in *View Options* or via the drop-down when using “Dynamic IsoSurface”.

Single IsoSurface at Value Negative Capped IsoSurface Positive Capped IsoSurface



- Added the ability when plotting a *Section Cut,* set to *Cut Model*, to choose if the display is showing all elements on the side of the plane toward the positive plane normal are removed (“0..Negative Cap”) or all elements on the side of the plane toward the negative plane normal are removed (“1..Positive Cap”). This options can be set via the drop-down in the *Section Cut Options* dialog box or when using “Dynamic Cutting Plane”.

Section Cut, Negative Cap Section Cut, Positive Cap



## Geometry Interfaces

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

| FEMAP Interface | Latest Supported Version |
|---|---|
| Parasolid | 32.0 |
| NX | 2019 (version 1872.15) |
| SolidEdge | 2020 |
| CATIA V5 | V5 R8 - V5-6 R2018 SP2 |
| ACIS | 2019 1.0.1 |
| JT | 10 |

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 (Previously NX Nastran)
- Nastran Interfaces (Simcenter and MSC/MD)
- ANSYS Interface
- ABAQUS Interface
- 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 v2020.1 changes

## Simcenter Nastran Interface (Previously NX Nastran)

- Added read/write support for BOLTFRC entry to create/export bolt preloads for multiple solution sequences.

### SOL 401 and SOL 402 Only

- Added *Restart Parameters (SOL 401 and SOL 402)* section to *Multi-Step Global Control Options* dialog box, which is now also available for SOL 401, and contains five options.

**Save Restart Data (RSTGEN)** - For the initial run, requests that restart data be saved for static, dynamic, and preload subcases. When disabled, which is the default, no restart data is saved, which writes nothing to the input file. When enabled, restart data is saved, which writes RSTGEN with a corresponding value of YES.

***Note:*** SOL 401 saves the restart data in the output OP2 file. SOL 402 saves the restart data into OP2 file, the SAMCEF SDB and ADB files, and the SAMCEF Output U18 file.

**Restart File Unit Number (SRTUNIT)** - For the restart run, defines the unit number of the external restart file. The unit number must also be specified on ASSIGN statement that references the physical file name of the *.op2 file. Must be greater than or equal to 161, Default = 161. Writes RSTUNIT field and corresponding value.

**Subcase to Restart From (SRTFROM)** - For the restart run, defines the subcase ID to restart from for the analysis. This subcase ID was defined in the initial run input file. Must be an integer value great than 1 and there is no default value, therefore it must be specified. Writes RSTFROM and corresponding value.

**Subcase to Execute From (EXEFROM)** - For the restart run, defines the subcase ID to execute from for the analysis. This subcase ID is defined in the restart run input file. Must be an integer value great than 1 and there is no default value, therefore it must be specified. Writes RSTFROM and corresponding value.

**Disable Model Validation fro Restart Run (MDLVAL)** - For the restart run, allows you to turn off the model validation check. When disabled, which is the default, nothing is written to the input file and the validation check is performed. When enabled, writes MDLVAL with a corresponding value of NO, which causes the validation check to not be performed.

### SOL 401 Only

- Added access to the *Multi-Step Global Control Options* dialog box. See SOL 401 and SOL 402 Only section.
- Added three options to the *Solution Parameters* section of the *Solution and Convergence Options* dialog box.

**Disable Mechanical Load (LOADOFF)** - Used to disable certain types of mechanical loads. Writes LOADOFF field with selected option (Default = "0..No Loads Turned Off").

"0..No Loads Turned Off" - No loads are disabled. Writes NONE.

"1..Body Loads Turned Off" - Disables body loads, which for Simcenter Nastran means any load being applied by a GRAV, RFORCE, RFORCE1, ACCEL, and/or ACCEL1 entry. Writes BODY.

“2..Boundary Loads Turned Off” - Disables boundary loads, which for Simcenter Nastran means any load being applied by a PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADE1, PLOADX1, FORCE, FORCE1, FORCE2, MOMENT, MOMENT1, MOMENT2, and/or SLOAD entry. Writes BOUNDRY.

“3..Both Boundary and Body Loads Turned Off” - Disables both body loads and boundary loads. Writes BOTH.

**Enable Inertia in Dynamics (INERTIA)** - Flag to include inertial effects in a subcase where *Analysis Type* is set to “3..Dynamics”. When enabled, which is the default, nothing is written. When disabled, writes INERTIA with corresponding NO field.

**Thermal Strain Loading (THRMST)** - Flag to include thermal strain in a static analysis. When enabled, which is the default, writes THRMST with corresponding YES field. When disabled, writes NO.

- Added three options to the *Stiffness Parameters* section of the *Solution and Convergence Options* dialog box.

**Quasi Newton-Raphson Iterations** - Only available when *Stiffness Update Strategy (KUPDATE)* is set to “2..Quasi Newton-Raphson”. Specifies the number of quasi Newton-Raphson iterations.

**Enable Stiffness Matrix Stabilization (MSTB)** - Flag to use matrix stabilization. When disabled, which is the default, writes nothing. When enabled, writes MSTAB with a value of 1.

**Matrix Stabilization Factor (MSFAC)** - Matrix Stabilization Factor. Default = 1.0E-10. Writes MSFAC field and corresponding value.

- Added four options to the *Convergence Parameters* section of the *Solution and Convergence Options* dialog box.

**Norm Criteria for Force (NORMP)** - Defines the norm criteria for the force convergence error function. Writes NORMP field with selected option (Default = “1..Norm 2 Criteria”).

“0..Norm 1 Criteria” - Use Norm 1 Criteria. Writes value of L1.

“1..Norm 2 Criteria” - Use Norm 2 Criteria. Writes value of L2.

“2..Norm Infinity Criteria” - Use Norm Infinity Criteria. Writes value of LINF.

**Norm Criteria for Displacement (NORMU)** - Defines the norm criteria for the displacement convergence error function. Writes NORMU field with selected option (Default = “1..Norm 2 Criteria”).

“0..Norm 1 Criteria” - Use Norm 1 Criteria. Writes value of L1.

“1..Norm 2 Criteria” - Use Norm 2 Criteria. Writes value of L2.

“2..Norm Infinity Criteria” - Use Norm Infinity Criteria. Writes value of LINF.

**Force Error Function Denominator (REFP)** - Optionally replaces the denominator in the force convergence error function. If set to -1.0, nothing is written. If entering a value, must be greater than 0.0. Writes REFP field and corresponding value.

**Disp. Error Function Denominator (REFU)** - Optionally replaces the denominator in the displacement convergence error function. If set to -1.0, nothing is written. If entering a value, must be greater than 0.0. Writes REFU field and corresponding value.

- Added one option to the *Time Integration* section of the *Solution and Convergence Options* dialog box.

**Modified Generalized Alpha Param (RHOINF)** - Modified generalized alpha method parameter, Rho Infinity. Writes RHOINF field and corresponding value. Value must be 0.0, 1.0, or any value in-between 0.0 and 1.0. Default = 0.8.

### SOL 402 Only

- Added two options to the newly created *Global Parameters (SOL 402)* section of the *Multi-Step Global Control Options* dialog box.

**Stress-Strain Measure for Output (STROUT)** - Choice of the stress-strain measure for the output of stress or strain simulation results. Writes STROUT with the selected option. If an option is specified for *Stress-Strain Measure for Plastic Matl.*, it will be the default for STROUT. Otherwise, the default is “1..Log Strain, Cauchy Stress” for large strains (PARAM,LGSTRN,1) or “2..Biot Strain, Biot Stress” for small strains (PARAM,LGSTRN,0).

“0..Green Strain, PK2 Stress” - writes a value of 0.

"1..Log Strain, Cauchy Stress" - writes a value of 1.

"2..Biot Strain, Biot Stress" - writes a value of 2.

"3..Log Strain, Kirchhoff Stress" - writes a value of 3

***Note:*** Results in metallic shell elements, multi-layered shell elements, and beam elements are always output in the using the "0..Green strain, PK2 stress" measure, regardless of the option set for *Stress-Strain Measure for Output (STROUT)*.

**Stress-Strain Conversion Method (STRCONV)** - Choice of the stress-strain curves conversion from Engineering to Cauchy. Writes STRCONV with the selected option (Default = "0..Exact Method").

"0..Exact Method" - writes a value of 0.

"1..Standard Method" - writes a value of 1.

***Note:*** See Simcenter Nastran Quick Reference Guide for information about the "Exact Method" and "Standard Method"

- Added three options to the newly created *Advanced Parameters (SOL 402)* section of the *Multi-Step Global Control Options* dialog box.

**Threshold for Zero Pivots (PRECPIVO)** - Threshold to detect zero pivots. Must be greater than 0.0, Default = 1.0E-8. Writes PRECPIVO field and corresponding value.

**Modes to Save as Output (NKINE)** - Number of kinematic modes to save as output. Must be greater than 0, Default = 10. Writes NKINE field and corresponding value.

**Version of Param. Defaults (MODEVERS)** - Allows the user to change the default behavior of parameters to be the same as an earlier version of Simcenter Nastran. Writes MODEVERS with corresponding field which specifies the version in a particular format. For instance, if "1..2019.1" is specified, writes a value of 20191. Default = "0..Default", which is the version of Simcenter Nastran which shipped with the version of FEMAP being used.

- Added two options to *Multi-Step Control Options* dialog box, two in *Time Stepping* and one in *Analysis Control*. Also, the *Other Options* section was divided into *Plasticity and Creep Control* and *Internal Restart*. In addition, the name of the *Solution/Convergence* button was changed to *Iteration and Control*.

**Activate Time Stepping Based on Disp/Temp Error (ERCD)** - Flag to activate time step criterion based on the absolute error of the displacement/temperature change. When disabled, which is the default, writes nothing. When enabled, writes ERCD with corresponding value of 1.

**Allowable Change/Step (PRED)** - Only available when *Activate Time Stepping Based on Disp/Temp Error (ERCD)* is enabled. Value of absolute allowable displacement/temperature change per time step. Must be above 0.0 and there is no default, therefore it must be specified to write PRED and corresponding value.

**Save Data at Beginning of Computation (IAR0)** - Specify if data must be saved at the beginning of the computation of a sequentially dependent subcase. When enabled, which is default, writes nothing and data is saved. When disabled, writes IAR0 with corresponding value of -1 and data is not saved.

- Added two options to *Solution and Convergence* dialog box, one in *Equilibrium and Convergence* and one in *Contact*. Also, removed *Time Step Criteria* and *Allowable Displacement (PRED)* from the *Analysis Options* section. In addition, updated layout of the entire dialog box to make more efficient use of space.

**Rigid Body Motion Convergence (OTRE)** - Choice to allow an easier convergence when no external loads are applied (rigid body motion). Writes PLAS with the selected option (Default = "1..Enable based on PRCE, PRCQ, and PRCR criteria").

"0..Disable Convergence" - Writes value of 0.

"1..Enable based on PRCE, PRCQ, and PRCR criteria" - Writes value of 1.

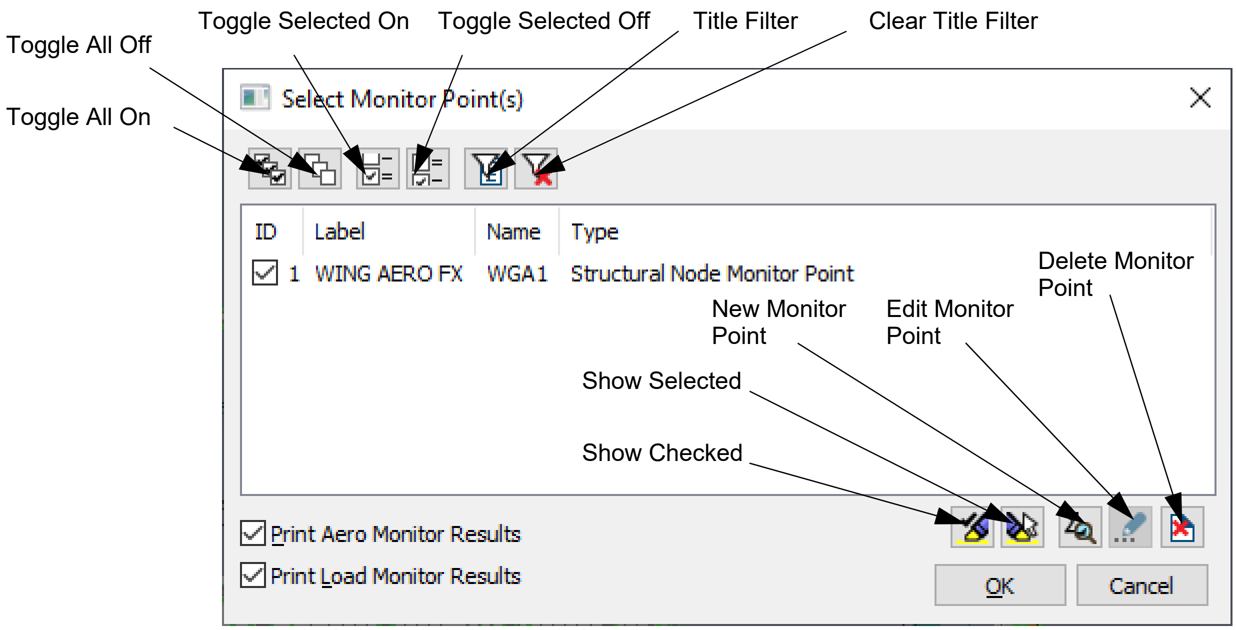
"2..Enable based on PRCE, PRCQ, PRCR, and REFP" - Writes value of 2.

**Threshold for Contact Force Variation (PRCF)** - Precision threshold for the contact forces variation. This value can be compared to TESC value in the *.f06 file.Must be greater than 0.0, Default = 0.1. Writes PRCF field and corresponding value.

### *Nastran Interfaces (Simcenter and MSC/MD)*

- Added support for “Large IDs” (i.e., IDs above 99,999,999). Any entity with a “Large ID” will automatically be written in large field format. It up to the user to confirm that any type of entity which has been assigned a “Large ID” is allowed to have a “Large ID” for the specified solution sequence in the specified solver.
- Added *Monitor Points* item in *Analysis Set Manager* for a number of different analysis types, which provides access to the *Select Monitor Point(s)* dialog box.

Highlighting the *Direct Matrix Input* item, then clicking *Edit* opens *Select Monitor Point(s)* dialog box:




Existing monitor points are shown with their *Label*, *Name*, and *Type*. Only the monitor point(s) which have a check box toggled “on” will be written to the Nastran input file. Simply check the monitor points to include. It is also possible to highlight any number of monitor points from the list. Holding down the “Ctrl” key will enable highlighting of multiple monitor points. Holding down “Shift” while picking a “first” row, then a “last” row, will highlight a range of monitor points. Once highlighting is complete, use the *Toggle Selected On* or *Toggle Selected Off* icon buttons to check or uncheck the highlighted monitor point(s).

Click the *Title Filter* icon button, then enter text into the *Title Includes* field to reduce the list to only include monitor points whose Titles include that text. Click *Clear Filter* icon button to return all monitor points to the list.

The *Toggle All On* button chooses all monitor points currently in the list, so if the list is “filtered”, only those monitor points will be selected. Click *Toggle All Off* to have no monitor points selected in the list.

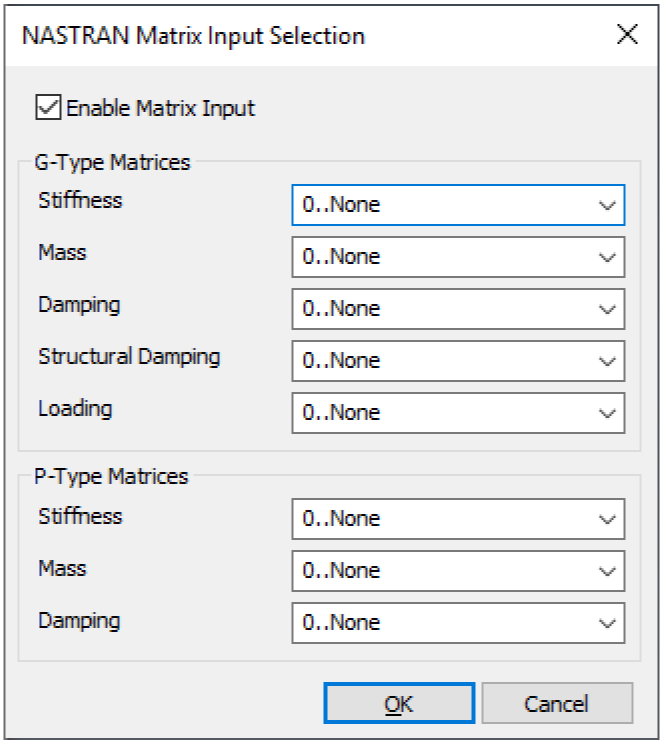
The *Show Checked* and *Show Selected* icon buttons are used to highlight the included entities for each monitor point in the graphics window. Using *Show Checked* will highlight the included entities for each monitor point checked in the *Select Monitor Point(s)* dialog box, while *Show Selected* uses the monitor points currently highlighted in the *Select Monitor Point(s)* dialog box.

The *New Monitor Point* icon button can be used at any time to display the *NASTRAN Monitor Point Manager* dialog box, which can then be used to create either type of monitor point, while the *Edit Monitor Point* icon button can be used to edit the monitor point(s) currently highlighted in the *Select Monitor Point(s)* dialog box. The *Delete Monitor Point* icon button can be used to delete the monitor point(s) currently highlighted.

The *Print Aero Monitor Results* and *Print Load Monitor Results* options are used to control if and how many times the MONITOR entry is written to case control. For Simcenter Nastran, neither of these options is available for Static Aeroelasticity (SOL 144) and only *Print Load Monitor Results* is available in the Master Case, or any subcase, for Linear Statics (SOL 101) or Normal Modes/Eigenvalue (SOL 103). For MSC Nastran, both requests are available, but only in the Master Case for any supported analysis type.

- Added *Direct Matrix Input* item in *Analysis Set Manager* for a number of different analysis types, which provides access to the *NASTRAN Matrix Input Selection* dialog box.

Highlighting the *Direct Matrix Input* item in either the *Master Requests and Conditions* or in any subcase, then clicking *Edit* opens the *NASTRAN Matrix Input Selection* dialog box:




Check the “Enable Matrix Input” box for the analysis set or subcase to be able to select *G-Type Matrices* and/or *P-Type Matrices*, which will be written to either the Case Control section, either for the Master Case or a specific subcase. In order to have items available for selection, Direct Matrix Input entities must exist in the model. A “Single Matrix” or “Matrix Combination” can be selected for any drop-down in this dialog box.

If a “Single Matrix” is selected in a drop-down, the appropriate Case Control entry will be written, along with the *Name* specified for the direct matrix input entity. If a “Matrix Combination” is selected in a drop-down, the appropriate Case Control entry will be written, along with an equation consisting of scale factors and the name of each direct matrix entity referenced in the combination.

***Note:*** Only direct matrix input entities, “Single Matrix”, which have *Source Data* set to *Internalized Data* can be selected using the drop-downs. If using a “Matrix Combination”, *Source Data* can be set to either *External File* or *Internalized Data* for the referenced direct matrix input entities. Also, a “Matrix Combination” only requires a single direct matrix input entity to be in the *Referenced Inputs* section.

The *G-Type Matrices* section contains five drop-downs:

**Stiffness** - Writes K2GG Case Control entry

**Mass** - Writes M2GG Case Control entry

**Damping** - Writes K2GG Case Control entry

**Structural Damping** - Writes K42GG Case Control entry

**Loading** - Writes P2G Case Control entry

The *P-Type Matrices* section contains three drop-downs:

**Stiffness** - Writes K2PP Case Control entry

**Mass** - Writes M2PP Case Control entry

**Damping** - Writes B2PP Case Control entry

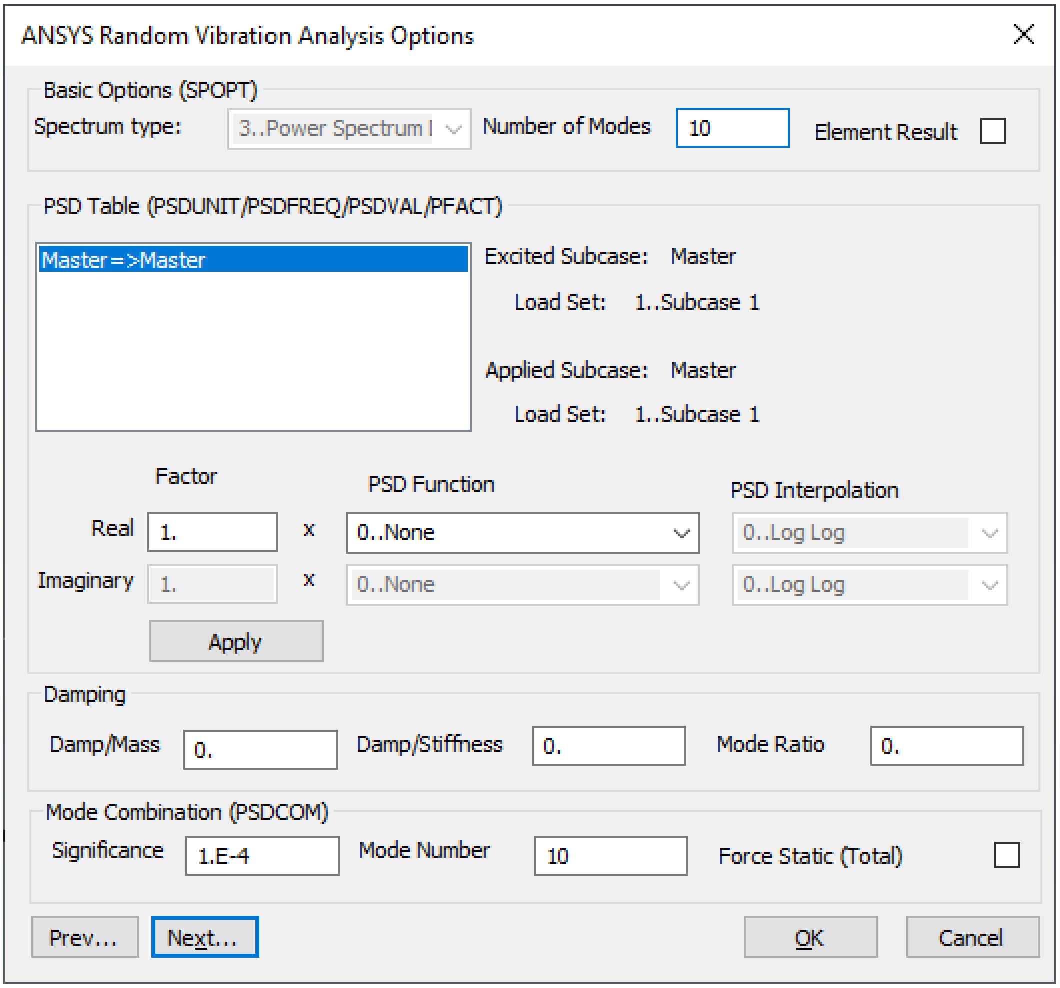
A number of bugs were corrected

For details, see Section 8, “Analysis Program Interfaces” in the *FEMAP User Guide*.

## *ANSYS Interface*

- Updated dialog box used for specification of PSD information and other options for random response analysis.

The *ANSYS Random Vibration Analysis Options* dialog box is used to specify options for frequency response analysis:




### Basic Options (SPOPT)

The options in this section specify a the Spectrum Type and other overall options for random response or spectrum analysis. Writes the SPOPT entry with appropriate names, values, etc for the current case. For detailed information, see ANSYS documentation for SPOPT.

**Spectrum Type** - Specifies the type of spectrum for the analysis. Currently, the only available option is “3..Power Spectral Density”, but other options exist, which may be supported in a future release.

- 0..Single-point Response Spectrum - Writes SPRS to “Sptype” field on SPOPT
- 1..Multi-point Response Spectrum - Writes MPRS to “Sptype” field on SPOPT
- 2..Dynamic Design Analysis Method - Writes DDAM to “Sptype” field on SPOPT
- 3..Power Spectral Density - Writes PSD to “Sptype” field on SPOPT

**Number of Modes** - Specifies to use only the first “number of modes” from the modal analysis (i.e., value of “5” means to only use the first 5 modes from the modal analysis). Writes integer value to NMODE field on SPOPT. Maximum value allowed is 10,000.

**Element Result** - Element Results Calculation Key. When disabled, which is the default, writes NO to Elcalc field on SPOPT. When enabled, writes YES to Ecalc field.

### PSD Table (PSDUNIT/PSDFREQ/PSDVAL/PFACT)

The options in this section specify various options for the PSD function. For detailed information, see ANSYS documentation for PSDUNIT, PSDFREQ, PSDVAL and PFACT.

The numbers of the "Correlation Table" correspond to the cases that define the loads for your analysis. Where the cases listed are the same (i.e, Master=>Master or 1=>1), pick the case from the table, then enter a real factor and select a PSD function to apply to the case.

Once *Apply* is clicked to update the "Correlation Table" when the cases are the same (i.e., Master=>Master,1=>1), the information will appear as follows:

"Real" Factor*("Real" Function ID) + :PSD Interpolation option ("Real")

When the cases are different (such as 1=>2), you must decide whether you want to correlate the cases. If you don't enter any values for real and imaginary values, the cases will not be correlated. If you do want to correlate the cases, enter the real and imaginary factors and PSD functions.

Once *Apply* is clicked to update the "Correlation Table" when the cases are not the same (i.e.,1=>2), the information will appear as follows:

"Real" Factor*("Real" Function ID) + "Imaginary" Factor*("Imaginary" Function ID) :PSD Interpolation option ("Real") :PSD Interpolation option ("Imaginary")

### Damping

The values can be used to specify damping options for random response analysis.

**Damp/Mass** - Specifies Mass Matrix multiplier, Alpha, for damping to form the viscous damping matrix [C]. Writes the ALPHAD entry with value.

**Damp/Stiffness** - Specifies Stiffness Matrix multiplier, Beta, for damping to form the viscous damping matrix [C]. Writes the BETAD entry with value.

**Mode Ratio** - Defines constant modal damping ratio. Writes DMPRAT entry with value.

### Mode Combination (PSDCOM)

The values are to specify the power spectral density (PSD) mode combination method and additional values and writes PSDCOM entry.

**Significance** - Specifies a threshold value to be considered for mode combination. Only modes which have a significance level above this threshold, which is determined by the modal covariance matrix term divided by the maximum modal covariance term, will be combined. If a mode has a significance level below this threshold, it is considered insignificant and not contributing the modal combination. The higher the value, the less modes will be used. If set to 0.0, all modes will be combined. Writes the value to the SIGNIF field on PSDCOM entry.

**Mode Number** - Specifies to use only the first "number of modes" to be combined (i.e., value of "5" means to only use the first 5 modes for mode combination). Value must always be less than or equal to *Number of Modes* value in *Basic Options (SPOPT)* section. Writes integer value to COMODE field on PSDCOM.

**Force Static (Total)** - Identifies the forces to be combined. When disabled, which is the default, combines both the modal static and inertial forces (writes TOTAL to ForceType field of PSDCOM entry). When enabled, combines only the model static forces (writes STATIC to ForceType field of PSDCOM entry).

***Note:*** Some Ansys random vibration calculations, such as nodal results spectrum (RPSD), are performed in ANSYS post processors. Those post processing functions are not supported in FEMAP.

- Added *Portion of Model to Write* option to *Model Control* section of *ANSYS Command and Model Control* dialog box.Allows you to select a previously defined group, then only exports the supported entities in that group to the ANSYS input file. In some cases, exporting elements without associated nodes or material/property entries may be desired, but this creates an input file which cannot be run by ANSYS unless additional entities are added to the file.

***Note:*** If a "ready to run" input file is desired, it may be helpful to use the *Group, Operations, Add Related Entities* command on the group before selected with this option.

- Improved support for shell elements with variable shell thicknesses defined via *DIM and *SET, SECFUN.

**Note:** Shell thickness is defined at nodes, so if the thickness is discontinuous across shell elements, the translation will likely not be exact.

- Improved support for ESEL and NSEL entries by creating FEMAP Groups when these are encountered. Also, Groups in FEMAP can be used to write ESEL and NSEL entries.
- Improved support for loads and constraints using ANSYS element or node components

A number of bugs were corrected.

For details, see Section 8, “Analysis Program Interfaces” in the *FEMAP User Guide*.

### ABAQUS Interface

- Added read/write support for DLOAD, TRVEC to create/export directional pressure loads.
- Added support to read *CONNECTOR SECTION types BUSHING and BEAM. For type = BUSHING, *Spring/Damper* elements which reference a corresponding property with *Type* set to CBUSH are created. For type = BEAM, *Spring/Damper* elements which reference a corresponding property with *Type* set to CBUSH, with very large stiffness values in all directions, are created.
- Added support for reading CONTACT PAIR, SURFACE INTERATION, and SURFACE NAMEs and retaining them in FEMAP.
- Added support for reading off-axis terms from ROTARY INERTIA entries, which are stored as *Mass* elements in FEMAP.
- Updated import to read a maximum of 9 digits for ID fields in the ABAQUS input file.
- Removed support for import of fixed field format files, as they are now incompatible with solver.

A number of bugs were corrected.

For details, see Section 8, “Analysis Program Interfaces” in the *FEMAP User Guide*.

### LS-DYNA Interface

- Added *Portion of Model to Write* option to *LS-DYNA Model Options* dialog box. Allows you to select a previously defined group, then only exports the supported entities in that group to the LS-DYNA input file. In some cases, exporting elements without associated nodes or material/property entries may be desired, but this creates an input file which cannot be run by LS-DYNA unless additional entities are added to the file.

**Note:** If a “ready to run” input file is desired, it may be helpful to use the *Group, Operations, Add Related Entities* command on the group before selected with this option.

- Added support for Beam type “13..Timoshenko”. In FEMAP, this is set using an element formulation.
- Improved support for shell elements with variable shell thicknesses defined using the Variable fields on the *ELEMENT_SHELL_THICKNESS entry.

A number of bugs were corrected.

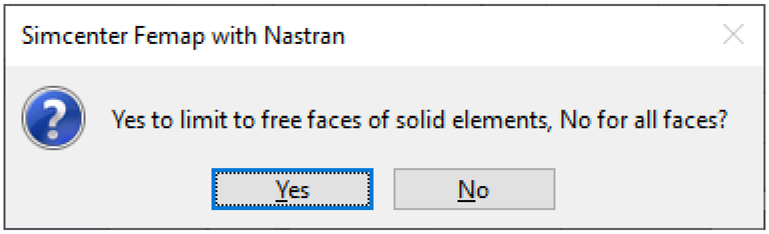
For details, see Section 8, “Analysis Program Interfaces” in the *FEMAP User Guide*.

## Tools

- Added the *Tools, Check, Mesh Interference* command.

This command examines a selected set of elements to determine if any of the selected elements interferes (i.e., element passes through and/or overlaps another element) with any of the other selected elements. This command can be used to determine if interference is occurring between line, planar, and/or solid elements and the cross-section of line elements and the thickness of planar elements are taken into consideration.

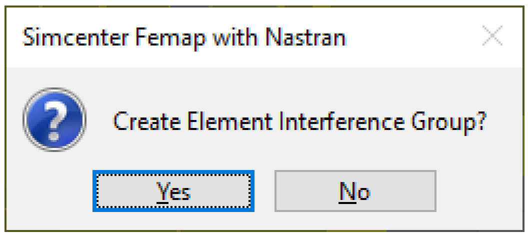
You first select the elements to be checked, then answer up to two additional questions. If any solid elements are selected, the following question will appear first:




Select *Yes* to only use the “free faces” of the selected solid elements or *No* to use all faces of solid elements.

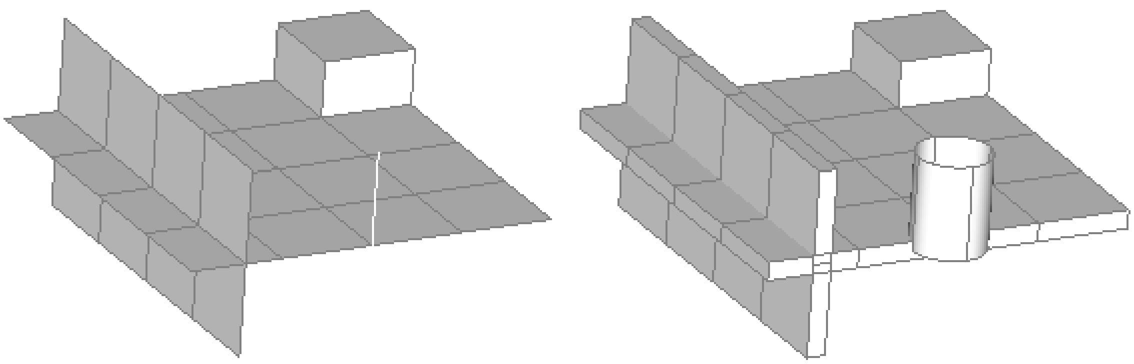
Any elements which interfere with each other will be highlighted in the graphics window using the current settings for the *Window, Show Entities* command.

In all cases where elements are found to be interfering with one another, the following question will appear:

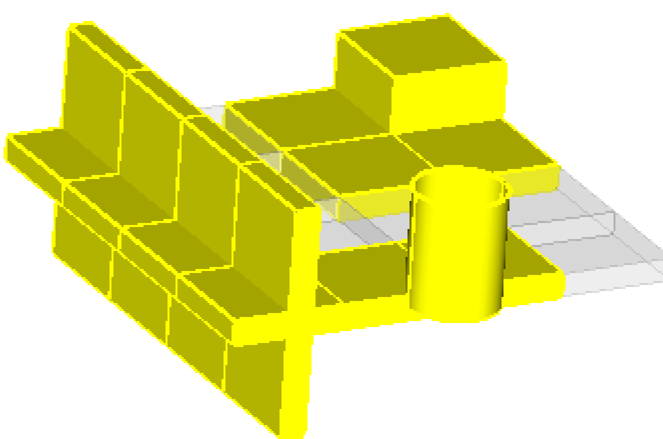



Select *Yes* to automatically create a group with all elements which interfere with at least one other element.

For example, this model (shown with and without cross-sections displayed):



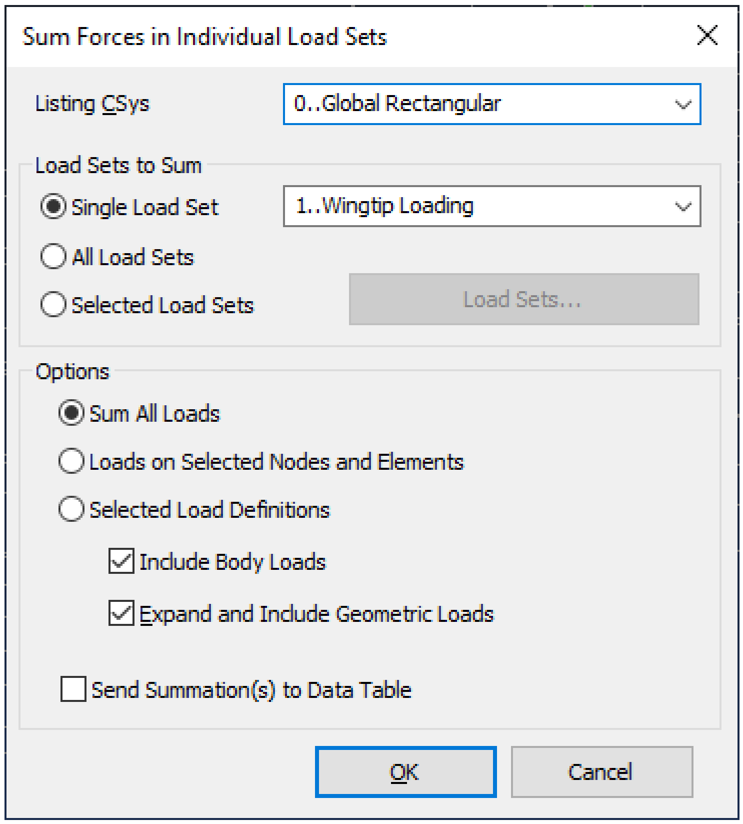
Would find that the highlighted elements interfere with at least one other element:



- Updated the *Tools, Check, Coincident Nodes* command by adding the *Delete Midside Nodes in Linear/Parabolic Transitions* option, which will delete any midside nodes on parabolic elements which are transitioning to linear elements after the corners of the elements have been merged.
- Updated the *Tools, Check, Sum Forces* command to be able to list the individual load summation for *All Load Sets* or any number of *Selected Load Sets*. In addition, added an option to send the individual summation for each load set to the *Data Table*.

This command checks the total forces about a selected location. Nodal forces, nodal moments, elemental pressures, and body loads from each selected load set are all used in this calculation.

The command will first ask the user to specify a location. The *Sum Forces in Individual Load Sets* dialog box will then appear:




The *Listing CSys* allows selection of an existing coordinate system to use for calculation.

The *Load Sets to Sum* section allows selection of a *Single Load Set* (by default, will be set to the “Active” Load Set), *All Load Sets*, or *Selected Load Sets* (Click *Load Sets* button to select Load Sets).

The *Options* section allows more control of the calculation. By default, the *Sum All Loads* option will be selected, with both *Include Body Loads* and *Expand and Include Geometric Loads* turned “on”. In this case, the calculation will include all the nodes and elements in the model, any geometry-based loads which create forces and pressures, and any body loads in each selected load set.

Choose *Loads on Selected Nodes and Elements* to instead select a subset of nodes and elements for the calculation.

Available only when *Load Sets to Sum* is set to *Single Load Set*, the *Selected Load Definitions* option can be used to only consider selected load definitions from the load set specified in the drop-down control for the calculation.

The *Include Body Loads* and/or *Expand and Include Geometric Loads* can be turned off to not include body loads and/or geometry-based loads, respectively and can be enabled/disabled when using any of the other options.

The *Send Summation(s) to Data Table* option can be used to send the calculated values to the *Data Table*.

**Note:** When using the *Send Summation(s) to Data Table* option, the calculated values for each load set will be written to a single row in the *Data Table*, which differs from the typical listing of these values to the active List Destination(s), which is described below.

When computing total effective moment, the specified location will be used to find the force couples caused by the applied forces, pressures, and body loads. A report in the following format will be produced. It will be written to the active List Destination(s).

```
Messages                                                                              x

Check Sum of Forces

Summation of Forces, Moments, Pressures and Body Loads for Set 1 (CSys 0)
  Nodal Force                FX =             0.  FY =        1500.  FZ =            0.
  Nodal Moment               MX =             0.  MY =           0.  MZ =            0.
  Pressure Force             FX =             0.  FY =           0.  FZ =            0.
  Body Translational Accel   FX =             0.  FY =           0.  FZ =            0.
  Body Varying Trans Accel   FX =             0.  FY =           0.  FZ =            0.
  Body Rotational Accel      FX =             0.  FY =           0.  FZ =            0.
  Body Rotational Velocity   FX =             0.  FY =           0.  FZ =            0.

Totals (CSys 0)
  About Location              X =        8.41162   Y =       -0.045   Z =      20.0216
  Forces                     FX =             0.  FY =        1500.  FZ =            0.
  Moments                    MX =      -104967.6  MY =           0.  MZ =     10636.32
```

In the first section of this report, all loads are simply added in global rectangular coordinates. The pressure force is determined by multiplying the elemental pressures defined by the element face area. This force is then applied in the direction opposite the face normal, at the centroid of the face.

To determine the forces created by the various body loads, the following is done:

For each element, the volume of the element is calculated then multiplied by the material currently assigned to the element to determine each element's mass. The calculated mass is then treated as a lumped mass located at each element's centroid. The individual lumped masses are multiplied by the accelerations or rotational velocities to determine a force at the element centroid, all of which are then summed to determine the overall force load for each type of body load.

**Note:** The approach used for calculating forces from body loads is only accurate to compute rotational loads if the effect of the inertias of the individual elements are small compared to the effects of the point masses (i.e. the model contains many small elements).

The *Totals* section of the report shows the specified location. The total forces will include the nodal force, the pressure force, and the body load force(s). The total *Moments* will include nodal moments, and the force couples produced by the nodal forces, pressure forces, and body load forces around the specified location.

## Model Merge

- Added support for Analysis Studies and Tables to be considered by the *File, Merge* command.

## OLE/COM API

To minimize the immediate impact on users with APIs, all existing API properties and Methods will CONTINUE TO WORK as before, BUT will only work when using the pre-2020.1 output vector IDs. This means existing APIs will works as they always have, and can be updated at your convenience to the new output vector IDs.

**Note:** While API scripts will work as they did before, if an API creates an output vector, and the ID of that output vector has changed, then the new ID for that output vector will be displayed in the FEMAP User Interface. If the pre-2020.1 output vector ID is used later in the API, it will NOT be found.

A large number of new API methods and properties which are specifically designed to work with the new Output Vector ID ranges have been added to Femap 2020.1. The names of these new property and method and appended with “V2”(ie NextVectorResetV2, VectorExistsV2, VectorsV2 etc..).

Supporting methods have also been added to convert output vector IDs back and forth from the two conventions:

- feOutputConvertV2019VectorIDToV2020V2 - allows users to update old APIs to use new output vector IDs
- feOutputConvertV2020VectorIDToV2019V2 - allows users to convert new output vector IDs to pre-2020.1 output vector IDs

**API Objects deprecated because all functionality for working with Output has been added to the Results Browsing Object, which is more robust**

- Removed Output Object and all related methods.

**Functions added specifically to work with new Output Vector ID Ranges (All new functions are appended with “V2”, even if the function did not exist prior to FEMAP 2020.1)**

- feFileAttachSaveV2
- feFileAttachSave2V2
- feFileWriteFNOV2
- feFileWriteFNO2V2
- feFileWriteFNO3V2
- feModelFileVersionV2
- feMeasureDistanceBetweenNodesV2
- feMeasureDistanceBetweenNodes2V2
- feMeasureAngleBetweenNodesV2
- feOutputProcessV2
- feOutputProcessCopyV2
- feOutputProcessMergeV2
- feOutputProcessLinearCombinationV2
- feOutputProcessRSSCombinationV2
- feOutputProcessConvertV2
- feOutputProcessEnvelopeV2
- feOutputProcessEnvelopeFromSetsV2
- feOutputProcessErrorEstimateV2
- feOutputTransform2V2
- feOutputConvertV92VectorIDV2
- feOutputConvertV2019VectorIDToV2020V2
- feOutputConvertV2020VectorIDToV2019V2
- feOutputCompareV2
- feResultsToDataTableV2
- feResultsToDataTable2V2
- feResultsRankingToDataTableV2
- feOutputGlobalPlyV2
- feDeleteOutputV2
- feDeleteOutput2V2

- feDeleteOutputEntryV2
- feSelectOutputV2
- feSelectOutput2V2

**Attributes added specifically to work with new Output Vector ID Ranges (All new attributes are appended with “V2”, even if the attribute did not exist prior to FEMAP 2020.1)**

- Added OutputVectorV2 and OutputVector2V2 attributes to the Chart Data Series Object
- Added VectorIDV2 attribute to the Optimization Responses (Limits) Object
- Added ContourVector1V2, ContourVector2V2, ContourVector3V2, ContourDataV2, ContourVecAddlV2, DeformDataV2, TraceIDV2, ContourVecID1V2, vContourVecID1V2, ContourVecID2V2, vContourVecID2V2, ContourVecID3V2, vContourVecID3V2, XYOutputDataV2, and vXYOutputDataV2 attributes to the View Object

**Methods added specifically to work with new Output Vector ID Ranges (All new methods are appended with “V2”, even if the method did not exist prior to FEMAP 2020.1)**

- Added AddAllInRangeInSetV2, AddAllTitleV2, AddComponentOutputVectorsV2, AddSimilarOutputVectorsV2, AddComplexOutputVectorsV2, SelectIDInSetV2, SelectMultiIDV2, and SelectOutputVectorIDV2 methods to the Set Object
- Added FindColumnV2, AddColumnV2 AddConversionColumnV2, GetColumnVectorV2, and VectorID methods to the Results Browsing Object Browsing Methods.
- Added NextVectorResetV2, NextVectorV2, VectorExistsV2, VectorInfoV2, VectorTitleV2, VectorComponentsV2, VectorLocationV2, VectorEntitiesV2, VectorsV2, VectorTitlesV2, and NonExistingUserVectorV2 methods to the Results Browsing Object Vector Methods.
- Added EntityValueV2 method to the Results Browsing Object Entity Value Methods.
- Added AddColumnV2, AddConversionColumnV2, FindColumnV2, and GetColumnVectorV2 methods to the Results Browsing Object Bulk Data Review Methods.
- Added AddScalarAtNodeColumnV2, AddVectorAtNodeColumnsV2, AddScalarAtElemColumnV2, AddElemWithCornerColumnsV2, AddScalarAtBeamColumnsV2, SetColumnVectorV2, GetScalarAtNodeColumnV2, SetScalarAtNodeColumnV2, GetScalarAtElemColumnV2, SetScalarAtElemColumnV2, GetVectorAtNodeColumnsV2, SetVectorAtNodeColumnsV2, GetElemWithCornerColumnsV2, SetElemWithCornerColumnsV2, GetScalarAtBeamColumnsV2, SetScalarAtBeamColumnsV2, GetScalarAtNodeSetV2, GetScalarAtElemSetV2, GetVectorAtNodeSetV2, GetElemWithCornerSetV2, and GetScalarAtBeamSetV2 methods to the Results Browsing Object Update and Save Methods.
- Added AddOutputV2 method to the DataTable Object.
- Added VarOutputMapV2 method to the Data Surface Object.
- Added MapFromModelToSetV2, MapFromModelToSet2V2, and MapFromModelToLocationV2 methods to the MapOutput Object.
- Added SetupElemV2 and GetElemV2 methods to the Optimization Responses (Limits) Object
- Added VectorV2 and OutputVectorsV2 methods to the Output Set Object

**New and modified API Objects and Attributes**

- Added Results Vector ID Query (feResultsIDQuery) object to the API. The feResultsIDQuery object has no attributes, only methods.
- Added Monitor Point (feMonitorPoint) object to the API. Also, added color, layer, title, Name, type, defCSys, outCSys, NodeGroup, ElemGroup, xyz, vxyz, SumComponents, vSumComponents, SumContributions, vSumContributions, HasNodeList, HasElemList, HasPanelList, CompTitle, NodeSetID, ElemSetID, PanelSetID, HasAeroMeshList, AeroMeshSetID, x, y, and z attributes to the feMonitorPoint Object.
- Added Direct Matrix Input (feMatrixInput) object to the API. Also, added title, Name, InputForm, InputPrecision, OutputPrecision, ComplexForm, colCount, matrixDir, AssignForm, UnitID, FileInputForm, ScaleFactor, and DataSource attributes to the feMatrixInput Object.

- Added AutoPopulate, NextSetStartID, NextVectorStartID, and NextStudyStartID attributes to the Results Browsing Object.
- Added CopyActiveColor and CopyToActiveCSys attributes to the CopyTool Object.
- Added NasMsnlCntINACCN, NasMsnlCntINERTIA, NasMsnlCntLOADOFF, NasMsnlCntMSTAB, NasMsnlCntMSFAC, NasMsnlCntNORMP, NasMsnlCntNORMU, NasMsnlCntREFP, NasMsnlCntREFU, NasMsnlCntRHOINF, NasMsnlkCnt2IAR0, NasMsnlkCnt2OTRE, NasMsnlkCnt2PRCF, NasEnableDMIG, NasDMIGK2GG, NasDMIGM2GG, NasDMIGB2GG, NasDMIGK42GG, NasDMIGP2G, NasDMIGK2PP, NasDMIGM2PP, NasDMIGB2PP, NasCaseMonitorEnabled, NasCaseMonitorAeroEnabled, NasCaseMonitorLoadEnabled, NasMsnlCntQuasiNewtonIter, and NasCaseAlternateTextLocation attributes to the Analysis Case Object.
- Added NasMsNLKGlobalSTROUT, NasMsNLKGlobalSTRCONV, NasMsNLKGlobalPRECPIVO, NasMsNLKGlobalNKINE, NasMsNLKGlobalMODEVERS, NasMsNLKGlobalRSTGEN, NasMsNLKGlobalRSTUNIT, NasMsNLKGlobalRSTFROM, NasMsNLKGlobalEXEFROM, NasMsNLKGlobalMDLVAL, NasMsnlModalMassForm, NasMsnlCntINACCN, NasMsnlCntINERTIA, NasMsnlCntLOADOFF, NasMsnlCntMSTAB, NasMsnlCntMSFAC, NasMsnlCntNORMP, NasMsnlCntNORMU, NasMsnlCntREFP, NasMsnlCntREFU, NasMsnlCntRHOINF, NasMsnlkCnt2IAR0, NasMsnlkCnt2OTRE, NasMsnlkCnt2PRCF, NasCaseAlternateTextLocation, NasEnableDMIG, NasDMIGK2GG, NasDMIGM2GG, NasDMIGB2GG, NasDMIGK42GG, NasDMIGP2G, NasDMIGK2PP, NasDMIGM2PP, NasDMIGB2PP, NasMonitorEnabled, NasMptSetID, NasMonitorAeroEnabled, NasMonitorLoadEnabled, and NasMsnlCntQuasiNewtonIter attributes to the Analysis Manager Object.
- Added Disabled attribute to the Connection Region Object.
- Added PreloadType attribute to the LoadBolt Object.
- Added NumberOfMatrices attribute to the Node Object.
- Added SectionCapMode attribute to the View Object.
- Added DO_Loads_Option, DO_Loads_RoundToZero, DO_Loads_RoundToZeroValue, DO_Loads_BeforeDecimalSeparator, DO_Loads_MinimumExponentSize, DO_Loads_ExponentDigits, DO_Loads_SigFigs, DO_Loads_LeadingZeros, DO_Loads_TrailingZeros, DO_Loads_DecimalPlaces, DO_Loads_MaxDigits, DO_Loads_BaseExponent, DO_Loads_OriginalDigits, and DO_Loads_Exponent attributes to the View Object. These attributes are used by the “Load Vectors” option in *View Options* to control the display of digits in the graphics window for Load Vectors
- Added DO_BCs_Option, DO_BCs_RoundToZero, DO_BCs_RoundToZeroValue, DO_BCs_BeforeDecimalSeparator, DO_BCs_MinimumExponentSize, DO_BCs_ExponentDigits, DO_BCs_SigFigs, DO_BCs_LeadingZeros, DO_BCs_TrailingZeros, DO_BCs_DecimalPlaces, DO_BCs_MaxDigits, DO_BCs_BaseExponent, DO_BCs_OriginalDigits, and DO_BCs_Exponent attributes to the View Object. These attributes are used by the “Constraint” option in *View Options* to control the display of digits in the graphics window for Constraints (Non-Zero)
- Added DO_Deformed_Option, DO_Deformed_RoundToZero, DO_Deformed_RoundToZeroValue, DO_Deformed_BeforeDecimalSeparator, DO_Deformed_MinimumExponentSize, DO_Deformed_ExponentDigits, DO_Deformed_SigFigs, DO_Deformed_LeadingZeros, DO_Deformed_TrailingZeros, DO_Deformed_DecimalPlaces, DO_Deformed_MaxDigits, DO_Deformed_BaseExponent, DO_Deformed_OriginalDigits, and DO_Deformed_Exponent attributes to the View Object. These attributes are used by the “Deformed Model” option in *View Options* to control the display of digits in the graphics window for the model when deformations are being shown as vectors
- Added DO_Criteria_Option, DO_Criteria_RoundToZero, DO_Criteria_RoundToZeroValue, DO_Criteria_BeforeDecimalSeparator, DO_Criteria_MinimumExponentSize, DO_Criteria_ExponentDigits, DO_Criteria_SigFigs, DO_Criteria_LeadingZeros, DO_Criteria_TrailingZeros, DO_Criteria_DecimalPlaces, DO_Criteria_MaxDigits, DO_Criteria_BaseExponent, DO_Criteria_OriginalDigits, and DO_Criteria_Exponent attributes to the View Object. These attributes are used by the “Contour/Criteria Style” option in *View Options* to control the display of digits in the graphics window for values of a Criteria Plot, when Max/Min Labels are being displayed in a Contour Plot, and/or when Beam Diagrams are displayed with Labels

- Added DO_ContourLegend_Option, DO_ContourLegend_RoundToZero, DO_ContourLegend_RoundToZeroValue, DO_ContourLegend_BeforeDecimalSeparator, DO_ContourLegend_MinimumExponentSize, DO_ContourLegend_ExponentDigits, DO_ContourLegend_SigFigs, DO_ContourLegend_LeadingZeros, DO_ContourLegend_TrailingZeros, DO_ContourLegend_DecimalPlaces, DO_ContourLegend_MaxDigits, DO_ContourLegend_BaseExponent, DO_ContourLegend_OriginalDigits, and DO_ContourLegend_Exponent attributes to the View Object. These attributes are used by the "Contour/Criteria Legend" option in *View Options* to control the display of digits in the graphics window for the Contour Legend
- Added DO_ContourArrow_Option, DO_ContourArrow_RoundToZero, DO_ContourArrow_RoundToZeroValue, DO_ContourArrow_BeforeDecimalSeparator, DO_ContourArrow_MinimumExponentSize, DO_ContourArrow_ExponentDigits, DO_ContourArrow_SigFigs, DO_ContourArrow_LeadingZeros, DO_ContourArrow_TrailingZeros, DO_ContourArrow_DecimalPlaces, DO_ContourArrow_MaxDigits, DO_ContourArrow_BaseExponent, DO_ContourArrow_OriginalDigits, and DO_ContourArrow_Exponent attributes to the View Object. These attributes are used by the "Contour Arrow Options" option in *View Options* to control the display of digits in the graphics window for values of Contour Arrows
- Added DO_FreeBody_Option, DO_FreeBody_RoundToZero, DO_FreeBody_RoundToZeroValue, DO_FreeBody_BeforeDecimalSeparator, DO_FreeBody_MinimumExponentSize, DO_FreeBody_ExponentDigits, DO_FreeBody_SigFigs, DO_FreeBody_LeadingZeros, DO_FreeBody_TrailingZeros, DO_FreeBody_DecimalPlaces, DO_FreeBody_MaxDigits, DO_FreeBody_BaseExponent, DO_FreeBody_OriginalDigits, and DO_FreeBody_Exponent attributes to the View Object. These attributes are used by the "Freebody Vectors" option in *View Options* to control the display of digits in the graphics window for values of Freebody arrows
- Updated NodeMergeOption attribute on the Geometry Preparation and Meshing Object. It now has 4 possible values instead of 3.

### New and Updated API Methods

- Added Find, Nodal, NodalWithComponents, NodalOther, Line, LineAllLocations, LineOther, Plate, PlateWithCorners, PlateOther, Laminate, LaminateWithCorners, LaminateGlobalPly, LaminateGlobalPlyWithCorners, Solid, SolidWithCorners, SolidOther, LaminateSolid, LaminateSolidWithCorners, LaminateSolidGlobalPly, LaminateSolidGlobalPlyWithCorners, Elemental, Complex, and User methods to the Results Vector ID Query Object.
- Added GetComponentSet, ClearComponentSet, PutComponentSet, GetAeroMeshSet, PutAeroMeshSet, and ClearAeroMeshSet methods to the Monitor Point Object.
- Added GetMatrixEntry, PutMatrixEntry, GetCombination, PutCombination, GetMatrixList, PutMatrixList, GetCombinations, and PutCombinations methods to the Direct Matrix Input Object.
- Added SetRow, SetRows, SetRowsByID, SetColumn, SetColumns, and SetRowsAndColumnsByID methods to the Results Browsing Object Browsing Methods.
- Added NumberOfStudies, NextStudyReset, NextStudy, StudyExists, StudyInfo, StudyTitle, StudyNotes, and StudyCreate methods to the Results Browsing Object Study Methods.
- Added SetCreate method to the Results Browsing Object Set Methods.
- Added SetColumnInfo, GetColumnInfo, SetColumnsComponent, GetColumnComponents, SetColumnTitle, SetColumnInteger, GetColumnInteger, SaveColumn, Save, Sort, and GetColumnAbsMax methods to the Results Browsing Object Update and Save Methods.
- Added IncludeOtherEntities2 methods to the CopyTool Object.
- Added IncludeOtherEntities2 methods to the MoveTool Object.
- Added PutMptSetID, GetMptSet, ClearMptSet, PutMptSet methods to the Analysis Manager Object.
- Added PutArray, GetAllArray, and PutAllArray methods to the BCNode Object.
- Added InitAllLayerInfo, InitLayerContentsInfo, InitLayerGroupReferenceInfo, InitLayerViewReferenceInfo, ClearLayerInfo, FindLayerInfo, ResetNextLayerInfo, NextLayerInfo, FindAllReferencedLayers, FindEmptyLayers, and FindMissingLayers methods to the Layer Object.
- Added PutArray, GetAllArray, and PutAllArray methods to the LoadBolt Object.
- Added PutArray, GetAllArray, and PutAllArray methods to the LoadETemp Object.

- Added PutArray, GetAllArray, and PutAllArray methods to the LoadMesh Object.
- Added PutArray, GetAllArray, and PutAllArray methods to the LoadNTemp Object.

**The following functions have been added or updated:**

- feCheckElemInterference
- feConnectLinearParabolic
- feCheckCoincidentNode4
- feLicensePrintInfo
- feLicenseGetInfo
- feSplineMultipleCurves
- feMeshConnectRigid
- feModifyPlateThickOffsetBetweenNodes
- feModifyPlateThickOffsetValue
- feModifyPlateThickOffsetEquation
- feModifyPlateThickOffsetReset

**New and updated Global Variables**

- Added Pref_DL_UseIntlAll, Pref_DL_DecimalSeparator, Pref_DL_DigitGroupSeparator, Pref_DL_DigitGroupOption, Pref_DL_DigitGroupFlag, Pref_DL_ExponentChar, Pref_DL_TrailingZeros, Pref_DL_ForceSingleZero, Pref_DL_ForcePlusSign, and Pref_DL_NegativeBraces to set Local preferences for Digit Control.
- Added Pref_DO_Option, Pref_DO_RoundToZero, Pref_DO_RoundToZeroValue, Pref_DO_BeforeDecimalSeparator, Pref_DO_MinimumExponentSize, Pref_DO_ExponentDigits, Pref_DO_SigFigs, Pref_DO_LeadingZeros, Pref_DO_TrailingZeros, Pref_DO_DecimalPlaces, Pref_DO_MaxDigits, Pref_DO_BaseExponent, Pref_DO_OriginalDigits, and Pref_DO_Exponent to set Options preferences for Digit Control.
- Added Pref_SkipDMIGEntries to set preference on Interfaces tab.
- Added Pref_API_WarningLevel and Pref_API_WarningInDialog to set preferences for warning messages about deprecated API calls (can only be set in GUI when the warning dialog is displayed).
- Added SelectOutputSetsFromStudies, which when set to True, will show “Study View” instead of “All Results View” in dialog boxes which display it both ways, such as “Select Output Sets”.
- Added InitializationPath and ConfigurationPath, to set the directory path to the FEMAP Initialization file (femap.ini) or the FEMAP Configuration file (config.ini).
- Updated Pref_RenderTrailingZeroes to set the same value as Pref_DL_TrailingZeros

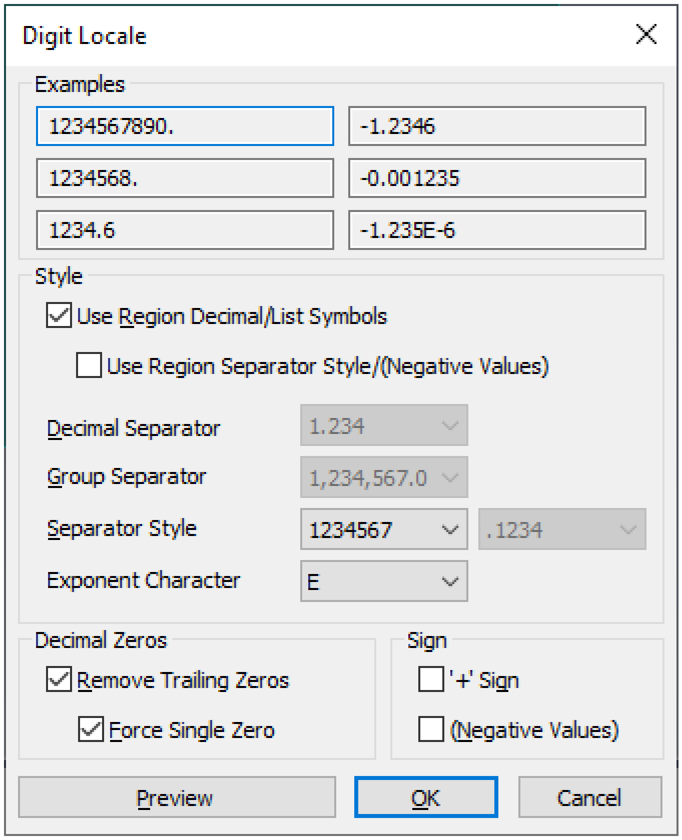
## *Preferences*

### *Graphics*

- Removed *Trailing Zeros* option from *Graphics Options* section, as this options is now handled by the settings in the *Digit Control* section.
- Added *Digit Control* section. This section contains two buttons, *Locale* and *Options...*, which are used to set global settings for the display format of digits for real number values in the graphics window.

#### *Locale button*

Opens the *Digit Locale* dialog box, which is used to specify overall options for *Style*, *Decimal Zeros*, and *Sign* for all digits being displayed in the graphics window




Pressing *Preview* will use the settings currently specified in the *Digit Locale* dialog box to temporarily modify the display of digits in the graphics window. Click *OK* to confirm settings or *Cancel* to leave settings unchanged.

### Examples

This section simply shows several examples of how digits will appear in the graphics window, based on the options selected in the lower portions of the dialog box.

### Style

The *Style* section contains two options near the top of the section which force FEMAP to follow certain settings for the display of digits in the graphics window specified at the Windows operating system level. Should one or both of these options be disabled, individual controls become available to control display of digits in the graphics window.

When *Use Region Decimal/List Symbols* is enabled, which is the default, the "region" is determined by querying the "Display Language" currently specified for the Windows operating system. When enabled, the character for *Decimal Separator* and character for *Group Separator* are automatically selected and cannot be changed via the individual controls. For example, in many European and South American countries, the "," is used to display real numbers and the "." used to separate different numbers in a list. When this option is disabled, the characters used for *Decimal Separator* and/or *Group Separator* can be specified individually by the user. Please see below.

When *Use Region Separator Style/(Negative Values)* is enabled, which is NOT the default, then the *Separator Style* option is also automatically by the "region" and all negative values are shown in parentheses instead using a negative sign. For example, a value of -123.456 would be shown as (123.456). When this option is disabled, the *Separator Style* can be manually selected from the drop-down and the *(Negative Sign)* option in the *Sign* section can be enabled by the user. This option is only available to enable when *Use Region Decimal/List Symbols* is also enabled.

The individual controls in the *Style* section control:

**Character used as *Decimal Separator*** - choose from Space, Decimal Point (.), Comma (,), or Apostrophe (')

**Character used as *Group Separator*** - available options include a specific subset of Spaces, Decimal Points, Commas, and Apostrophes, which does not include the character specified for *Decimal Separator*. For example, if *Decimal Separator* is set to Decimal Point, then the available options for *Group Separator* are restricted to Spaces, Commas, and Apostrophes.

**Options for *Separator Style*** - specify the overall usage and spacing of the *Group Separator* character to the left of the *Decimal Separator*. Options include: do not use *Group Separator* character (12345678), use character every 3 digits to the left (12,345,678), use character for only for first 3 digits to the left (12345,678), or use character after first 3 digits to the left, then use every 2 digits further to the left (1,23,45,678). When using any option other than do not use *Group Separator* character, an additional drop-down becomes available to specify if and how the *Group Separator* will be used to the right of the *Decimal Separator*.

**Character used as *Exponent Character*** - choose from “E”, “e”, “D”, “d”, “F”, or “f”.

### Decimal Zeros

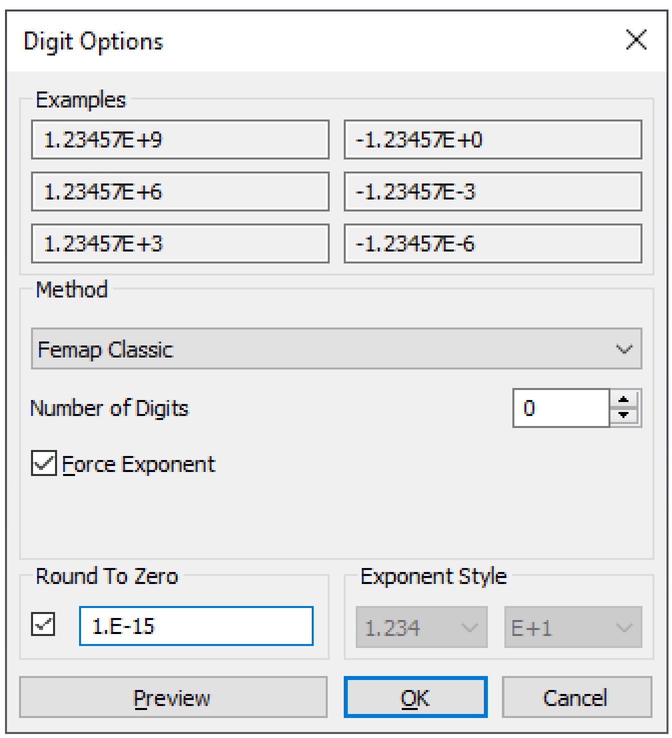
This section controls how zeros are displayed to the right of the *Decimal Separator*. Both options are enabled by default. When *Remove Trailing Zeros* is enabled, any value of 0 which ends a value will be removed (i.e., value of 123.45600 becomes 123.456). When *Force Single Zero* is enabled, a single 0 is added after the Decimal Separator, if there are a number of trailing zeros (i.e., value of 123.000 becomes 123.0).

### Sign

This section controls how the sign of digits are displayed in the graphics window. Both of these options are disabled by default. When *‘+’ Sign* is enabled, a positive sign (“+”) will appear in front of the value, meaning 123.456 will be displayed as +123.456. When *(Negative Value)* is enabled, negative values will be shown in parentheses instead of using a negative sign (“-”), meaning -123.456 will be displayed as (123.456).

## *Options button*

Opens the *Digit Options* dialog box, which is used to specify *Method*, *Round To Zero*, and *Exponent Style* for any option in *View, Options* which has *Method* in an individual *Digits Options* dialog box set to *Use Preferences*.




Pressing *Preview* will use the settings currently specified in the *Digit Locale* dialog box to temporarily modify the display of digits in the graphics window. Click *OK* to confirm settings or *Cancel* to exit the dialog box with settings unchanged.

The following options in *View, Options* contain a *Digits* button which can be used to override the options set in the *Digit Options* dialog box accessed via the *File, Preferences*:

***Labels, Entities and Color*** **Category** - *Load Vectors* and *Constraint*

***Tools and View Style*** **Category** - None

***PostProcessing*** **Category** - *Deformed Model*, *Contour/Criteria Style*, *Contour/Criteria Legend*, *Contour Arrow Options*, and *Freebody Vectors*

In a new model, the default *Method* for all of the above options in *View, Options* will be *Use Preferences*.

### Examples

This section simply shows several examples of how digits will appear in the graphics window, based on the options selected in the lower portions of the dialog box.

### Method

This section is used to select a *Method* to use as "global settings" for the display of digits in the graphics window. Each *Method* has additional options which can be used to further customize the display of digits. The *Methods* are:

***Femap Classic*** - Digits of real numbers are displayed in the same manner as they were in versions of FEMAP prior to 2020.1. The default value for *Digits* remains "0", which will also work as it did in previous versions. When *Number of Digits* is set to a value above "0", the number of displayed digits will attempt to follow the specified value, when possible. In addition, enabling the *Force Exponent* option will show the values as exponents (i.e., scientific notation).

For example, use a value of 12.456 and the *Force Exponent* option disabled. If *Number of Digits* set to:

- "0", value displayed as 12.456
- "1" or "2", value displayed as 12.
- "3", value displayed as 12.5
- "4", value displayed as 12.46
- "5" or above, value displayed as 12.456.

Use the same value of 12.456, with *Force Exponent* option enabled. If *Number of Digits* set to:

- "0", value displayed as 1.2456E+1
- "1", value displayed as 1.2E+1
- "2", value displayed as 1.25E+1
- "3", value displayed as 1.246E+1
- "4" or above, value displayed as 1.2456+1

***Exponent*** - Digits of real numbers are always displayed in exponent form. The *Number of Digits* controls the total number of digits visible to the left of the *Exponent Character* (by default, "E"). In addition, the options selected in *Exponent Style* are taken into account when using this method. Please see Exponent Style below.

For example, use a value of 12456.0. If *Exponent Style* options are set to the defaults and *Number of Digits* set to:

- "1", value displayed as 1.E+4
- "2", value displayed as 1.2E+4
- "3", value displayed as 1.25E+4
- "4", value displayed as 1.246E+4
- "5" or above, value displayed as 1.2456E+4

***Significant Figures*** - Digits of real numbers are displayed using the value specified for *Number of Significant Digits*. In some cases, this will cause certain values to appear in standard form, while others appear in exponent form. In addition, there are options for the *Maximum Zeros after decimal point* and *Maximum Zeros before decimal point*, which can be used to add or remove from before or after the *Decimal Separator*.

For Example, use values of 1234.5678 and 0.000123456. If *Maximum Zeros after Decimal Separator* is 0 and *Number of Significant Figures* is et to:

- “1”, values displayed as 1.E+3 and 1.E-4
- “2”, values displayed as 1.2E+3 and 1.2E-4
- “3”, values displayed as 1.23E+3 and 1.23E-4
- “4”, values displayed as 1235. and 1.235E-4
- “5”, values displayed as 1234.6 and 1.2346E-4
- “6”, values displayed as 1234.57 and 1.23456E-4 (will not change if *Number of Significant Digits* is increased)
- “7”, values displayed as 1235.568
- “8” and above, values displayed as 1234.5678

Use the same value of 0.000123456. If *Number of Significant Digits* is set to 6 and *Maximum Zeros after Decimal Separator* set to:

- “2” or below, value displayed as 1.23456E-4
- “3” or above, value displayed as 0.000123456

Use a value of 1230000. If *Number of Significant Digits* is set to 4 and *Maximum Zeros before Decimal Separator* set to:

- “2” or below, value displayed as 1.23E+6
- “3” or above, value displayed as 1230000.

***Decimal Places*** - Digits of real numbers are always displayed using the value specified for *Number of Decimal Places*. Because of this, the value specified for *Maximum Number of Digits* can potentially be used to move the *Decimal Separator* to the left, which will increase the value of the exponent, or to the right, which will decrease the value of the exponent. In addition, the value specified for *Base Exponent* can be used to specify a base value to the exponent which can potentially change the display of all values.

For Example, use a value of 123.4567. If *Maximum Number of Digits* is 8 and *Number of Decimal Places* set to:

- “1”, value displayed as 123.5
- “2”, value displayed as 123.46
- “3”, value displayed as 123.457
- “4”, value displayed as 123.4567
- “5”, value displayed as 123.45670
- “6”, value displayed as 12.345670E+1
- “7”, value displayed as 1.2345670E+2
- “8”, value displayed as 0.12345670E+3
- “9”, value displayed as 0.123456700E+3
- “16” (max), value displayed as 0.1234567000000000E+3

Use the same value of 123.4567. If *Number of Decimal Places* set to 5 and *Maximum Number of Digits* set to:

- “2” to “5”, value displayed as 0.12346E+3
- “6”, value displayed as 1.23457E+2
- “7”, value displayed as 12.34567E+1
- “8” or above, value displayed as 123.45670

Use the same value of 123.4567. If *Number of Decimal Places* set to 5, *Maximum Number of Digits* set to 8, and Base Exponent set to:

- “0” or below, value displayed as 123.45670
- “1”, value displayed as 12.34567E+1

- “2”, value displayed as 1.23457E+2
- “3”, value displayed as 0.12346E+3
- “4”, value displayed as 0.01235E+4
- “5”, value displayed as 0.00123E+5
- “6”, value displayed as 0.00012E+6
- “7”, value displayed as 0.00001E+7
- “8” or above, value displayed as 0.00000E+(value specified for *Base Exponent*)

#### Round to Zero

This option is used to specify a threshold for display of values. This option is enabled by default and has a default value of 1.E-15, meaning any value below 1.e-15 will be displayed using a value of 0.0 in the graphics window.

#### Exponent Style

This section is used to set options for the display of exponent values, before and after the *Exponent Character*.

The available options before the *Exponent Character* are to start the value with:

- Nothing before the *Decimal Separator* (i.e., .###)
- A zero before the *Decimal Separator* (i.e., 0.###)
- A single digit before the *Decimal Separator* (i.e., #.###)

The available options after the *Exponent Character* are to show the exponent:

- With nothing when exponent portion of value is E+0 (i.e., 1.234 would be shown as 1.234)
- Exponent with single digits, unless more are needed (i.e., E+1, E+10, E+100)
- Exponent with two digits, unless more are needed (i.e., E+02, E+12, E+102)
- Exponent with three digits, unless more are needed (i.e., E+003, E+013, E+103, E+1003)

### *Interfaces*

Added *Skip DMIG Bulk Data Entries* option to *Nastran Options* section. When this option is disabled, which is the default, DMIG entries encountered in the bulk data section of the Nastran input file will become Direct Matrix Input entities in FEMAP. When enabled, all DMIG entries encountered are skipped.

# What's New - version 2019.1 MP1

## Geometry Interfaces

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

| FEMAP Interface | Latest Supported Version |
|---|---|
| NX | 2019.1 v1872 |

For details, see "Geometry Interfaces" in the *FEMAP User Guide*.

## Analysis Program Interfaces

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

- Nastran Interfaces (NX and MSC/MD)
- ABAQUS Interface

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

### Nastran Interfaces (NX and MSC/MD)

- Added ability to automatically generate buckling subcases in FEMAP using the "MultiSet" functionality of the Analysis Set Manager. The generated subcases are identical to the synthetic Nastran buckling subcases that were exported in previous versions of FEMAP.

### ABAQUS Interface

- Added read support for *DISTRIBUTION and *DISTRIBUTION TABLE entries. Currently supported variable are ORIENTATION and ANGLE. ORIENTATION gets translated to material angle on the element and ANGLE becomes the layup angle for the ply. This will create property and layup for each laminate element, as compression or consolidation of layups is not performed.
- Added read support for *COUPLING and *DISTRIBUTING entries, which are translated to RBE3 elements when certain specifications are met on the entries. Independent nodes must be specified as *SURFACE using NODES, while Dependent (reference node) must be specified as a single node, although ABAQUS lets you specify a set (NSET) In addition, the COUPLING type must be CONTINUUM and the Weighting must be UNIFORM (at all nodes = 1.0 for all 6 components).

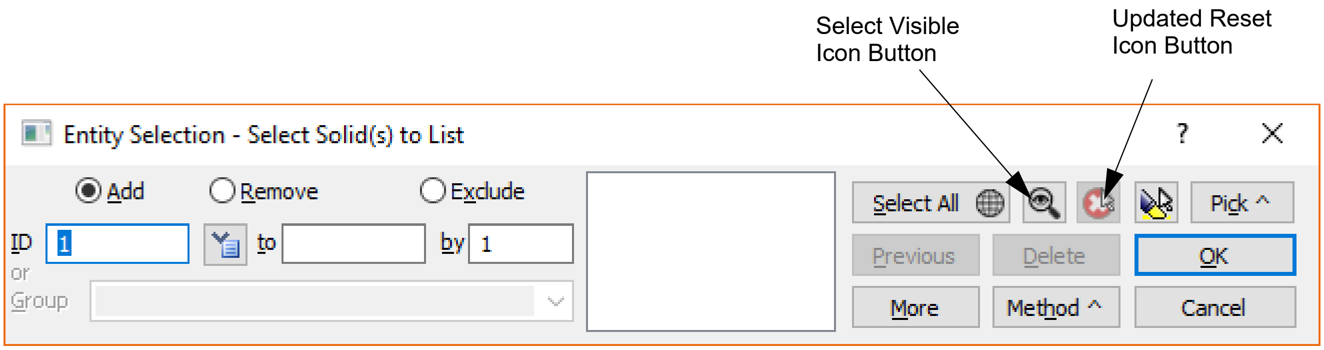
# What’s New - version 2019.1

## User Interface

General, Menu, Toolbars, Model Info tree, Meshing Toolbox, Charting pane, Entity Editor, and Data Table

### General

- Added *Select Visible* icon button to the standard entity selection dialog box. In addition, replaced *Reset* button with a *Reset* icon button (Red Circle with White “X”) and added an icon image to *Select All* button..




Using the *Select Visible* icon button will select all entities of the desired type which are currently “visible” in the active view. In this case, “Visible” means the entity, in the active view, is: A) not hidden by any setting for Visibility (i.e., overall entity type; individual entity; elements based on type, shape, or which reference hidden materials and/or properties); B) not hidden by the *Draw/Erase Toolbar* in any way; C) displayed based on the current *Group* display settings; and D) on a visible *Layer*.

**Note:** This includes any entities which are “visible”, but may not currently appear in the graphics window due to zooming, panning, or rotating in the view to only display a portion of the model.

- Added *Select Visible Only* option to the *Pick^* menu in the standard entity selection dialog box. When *Select Visible Only* is disabled, which is the default, any entity in the model which fit the criteria entered for a *By...* method or are related via an *Add...* option will be selected. When *Select Visible Only* is enabled, only entities which are currently visible in the active view and fit the criteria or are related will be selected. The methods on the *Pick^* menu which only consider “visible” entities when *Select Visible Only* is enabled are *By Size*, *By Color*, *By Model Data Value*, *By Output*, *Add Connected Tangent Curves*, *Add Connected Fillets*, *Add Tangent Surfaces*, *Add Connected Elements*, and *Add All Connected Elements*.

**Note:** When the *Select Visible Only* toggle is enabled, the icon next to the toggle will be highlighted and all items on the *Pick^* menu which follows the toggle will be appended with *(Visible Only)*.

- Added *using Orientation Node* to the *Method^* menu of the standard entity selection dialog box when selected elements, which selects any element which uses a selected node as an orientation node.

### Menu

- Added *Tools, Structural Analysis Toolkit* to the *Tools* menu. Simply opens the Structural Analysis Toolkit dockable pane when it is installed.
- Added *Modify, Renumber, View*; *Modify, Renumber, Chart*; and *Modify, Renumber, Chart Data Series* to the *Modify* menu. See section for more information. See Charting pane and Views for more information. Also, added *Modify, Update Elements, Solid Material CSys* and *Modify, Update Elements, Connect Linear/Parabolic Elements* to the *Modify* menu. See Elements section for more information.
- Added *Group, Element, Using Orientation Node* to the *Group* menu. See Groups and Layers section for more information.

- Updated the *List* menu, by replacing the *List, Output, Standard* and *List, Output, Use Format* commands with a single command, *List, Output, Formatted*. Also, updated the name of the *List, Output, Format* command to *List, Output, Custom Format Definition* and moved the command into the same section of the menu as *List, Output, Formatted*. See Listing section for more information.

### Toolbars

- Added *Locate in Model Info* toggle to *Selector Modes* menu of the *Select* toolbar. Only available when *Select Single Item* is the active setting in the *Selector Modes* icon menu. When *Locate in Model Info* mode is activated, the *Model Info* tree pane is currently visible, and the active entity is set to *Solid*, *CSys*, *Property*, *Material*, or *Layup*, the currently selected entity will also be highlighted in the *Model Info* tree.
- Added *Select Visible Only* toggle to *Selector Actions* menu of the *Select* toolbar. Allows you to select only from entities which are currently “visible” in the active view

***Note:*** “Visible” means the entity, in the active view, is A) not hidden by any setting for Visibility (i.e., overall entity type; individual entity; elements based on type, shape, or which reference hidden materials and/or properties); B) not hidden by the *Draw/Erase Toolbar* in any way; C) displayed based on the current *Group* display settings; and D) on a visible *Layer*.

- Added *Feature Lines* toggle to *View Style* menu of the *View* and *View - Simple* toolbars. Used enable and disable the display of feature lines, which are otherwise controlled by the *Feature Line* option in the *Tools and View Style* category of the *View Options* dialog box.

### Model Info tree

- Added *Copy* and *Renumber* commands on context-sensitive menu for *Views*, which are used to copy or renumber, respectively, any number of views currently highlighted in the *Model Info* tree.

### Meshing Toolbox

- Updated functionality of operations which update geometry, specifically ones which split existing geometry into smaller pieces, to attempt to automatically update geometry-based loads, constraints, and/or regions.

### Charting pane

- Added *Show When Selected* icon buttons to the *Data* tab for all data series types which allow selection of nodes and/or elements, as well as groups.
- Updated the location of certain fields and control on the *Data* tab for all data series types other than “0..Vector vs. Entity” and “5..Function”.
- Added ability to renumber *Charts* and *Chart Data Series* from the *Modify, Renumber* menu. Also, *Charts* and *Chart Data Series* can now be renumbered using the *Modify, Renumber, All* command.

### Entity Editor

- Added *Material CSys* item for Solid, Solid Laminates, and Solid Cohesive Elements which have a material coordinate system override (MATCID) assigned, which can be updated from the Entity Editor.

### Data Table

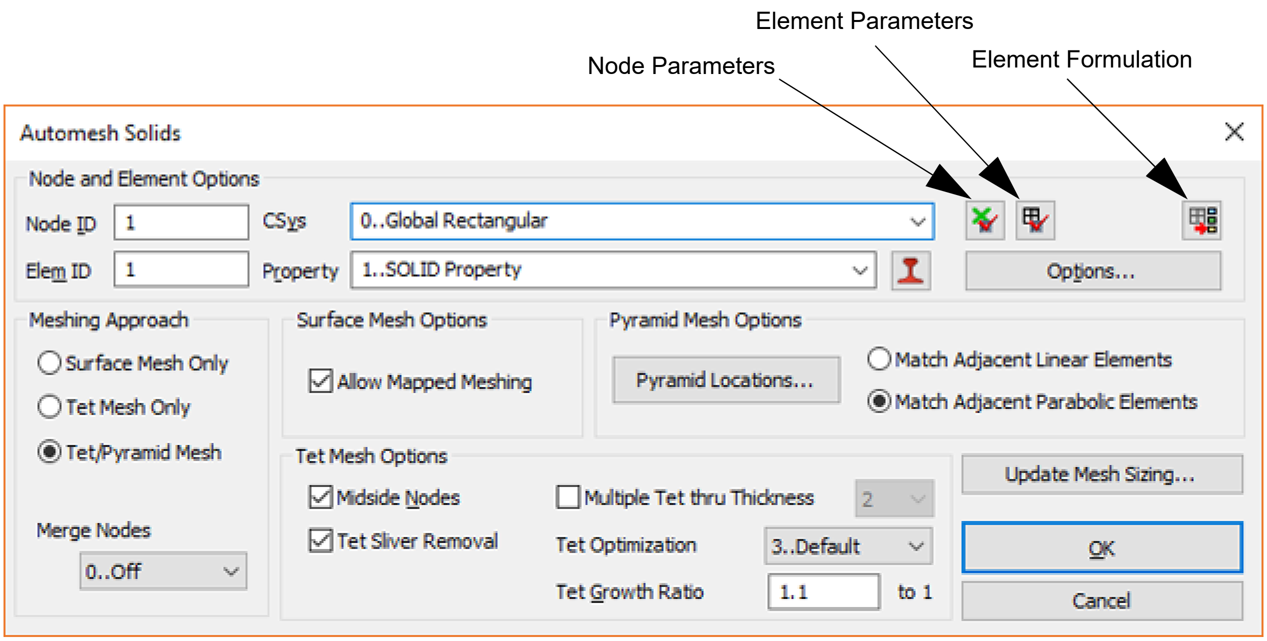
- Added *Previous On* icon buttons to the *Results to Add to Data Table* dialog box, which is accessed via *Add Output Columns* icon.
- Added *Material CSys* item for Solid, Solid Laminates, and Solid Cohesive Elements which have a material coordinate system override (MATCID) assigned.

## Geometry

- Updated *Modify, Project, Point* command by adding two options to the *Direction* section, *Radial Around Axis*, and *Radial Around Point*, which project points outwards from either a defined vector or specified location.
- Updated functionality of operations which update geometry, specifically ones which split existing geometry into smaller pieces, to attempt to automatically update geometry-based loads, constraints, and/or regions.

## Meshing

- Updated the *Mesh, Extrude, Element* and *Mesh, Extrude, Element Face* commands by adding the A*long Normal to Surfaces* option to the *Method* section of the *Generation Options* dialog box. This options is similar to using *Along Vector to Surfaces* when planar elements are selected, but uses each element's normal direction as the extrusion vector instead of requiring a vector to be specified.
- Updated all commands on the *Mesh, Extrude* and *Mesh, Revolve* menus by adding the *Extrude in Both Directions* option. This option works a little different for each method, but essentially it creates elements along or around the specified vector(s) as well as in the opposite direction(s).
- Updated the *Mesh, Geometry, Solids* command by adding the ability to automatically create pyramid elements in transition areas when *Meshing Approach* is set to *Tet/Pyramid Mesh*. In addition, added the *Pyramid Mesh Options* section to the dialog box to control certain aspects of the automatically generated pyramids.




The *Tet/Pyramid Mesh* option in the *Meshing Approach* section will create pyramid elements in a solid where any surface of a solid is:

...already meshed with quadrilateral elements, "linked" to a surface of another solid which was previously meshed with quadrilateral elements, "linked" to a surface of another solid which was previously meshed with hexahedral elements (bricks), and/or selected using the *Pyramid Locations* button in the *Pyramid Mesh Options* section of this dialog box.

Each quadrilateral element/element face will serve as the four-sided "base" of a pyramid element which extends from the surface "into" the solid, then the remainder of the solid is meshed with tetrahedral elements.

***Note:*** Typically, pyramids should only be used to transition from hexahedral elements to tetrahedral elements in portions of a model which cannot be properly subdivided for hex meshing.

These options in the *Pyramid Mesh Options* section allow the user to control different aspects of the pyramid elements automatically generated by the command. Each of these options are described below.

The *Pyramid Locations...* button displays the standard entity selection dialog box for surfaces and can be used to select surfaces, which have not been meshed, where pyramid elements should be created.

***Note:*** Typically, pyramids should only be used to transition from hexahedral elements to tetrahedral elements in portions of a model which cannot be properly subdivided for hex meshing.

The other option in this section controls if the four-sided faces of automatically generated pyramid elements will *Match Adjacent Linear Elements* (four-sided faces will not have midside nodes) or *Match Adjacent Parabolic Elements* (four-sided faces will have midside nodes). *Match Adjacent Parabolic Elements* is the default.

- Updated all commands on the *Mesh, Sweep* menu by adding a *Methods* section to the *Generation Options* dialog box. The *Along Curve* method allows these commands on the *Mesh, Sweep* menu to be used as they have existed in FEMAP for many releases, while the *Along Element Edges* method offers new functionality. The *Along Element Edges* method also introduces a new dialog box to facilitate the selection of element edges on both planar and solid elements. Below is an overview of the desired workflow for both methods, including a full description of the *Select Edges to Extrude Along* dialog box.

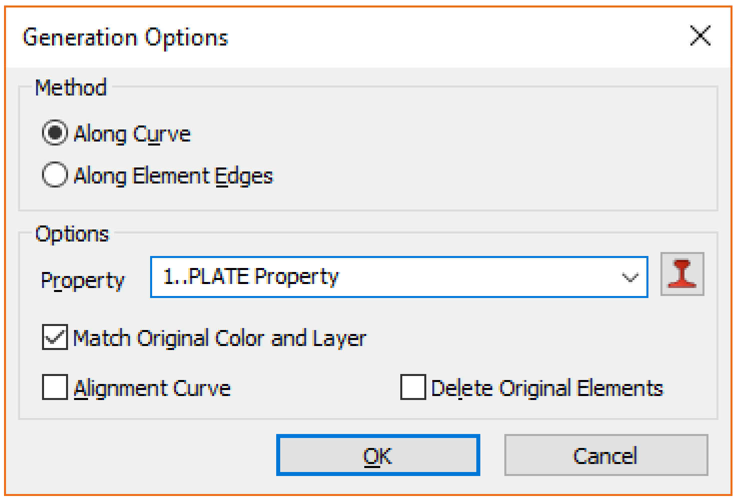
Use the following procedure to sweep curves and elements:

**1.** If planning to use the *Along Curve* options, define the mesh size using the *Mesh, Mesh Control, Size Along Curve* command for both the curve(s) to sweep and the curve(s) to sweep along, before using the *Sweep* command. The mesh size will determine the number of elements created for each curve.

**2.** Pick the *Mesh, Sweep, Curve*; *Mesh, Sweep, Element*; or *Mesh, Sweep, Element Face* command.

**3.** Select the curve(s) or element(s) to sweep using the standard entity selection dialog box.

**4.** Next, the *Generation Options* dialog box will appear. If using the *Along Curve* method, see “Step 7” for selection of curves to sweep along and, optionally, selection of an “alignment curve” or reference point (see “Step 6”). If using the *Along Element Edges* method, see “Step 8” for selection of element edges on a mapped mesh.




**5.** Pick the *Property* for the elements that will be created. For curves and line elements to sweep, pick a plane element property. For plane elements to sweep, pick a solid element property. When *Match Original Color and Layer* is enabled, which is the default, the color and layer of the original elements being swept will be used for the newly created elements, otherwise, the active element color and layer is used for the new elements. When *Delete Original Elements* is enabled, the original elements selected for sweeping will be deleted, otherwise, the original elements will be moved to the “end” of the newly swept elements, which is the default behavior.

**6.** If the curve to sweep along is planar, there is no need to pick any additional options. If the curve to sweep along is not planar, it is highly recommended to use the *Alignment Curve* option. The selection of the “alignment curve” will occur in the next step, after the selection of the curve(s) to sweep along.

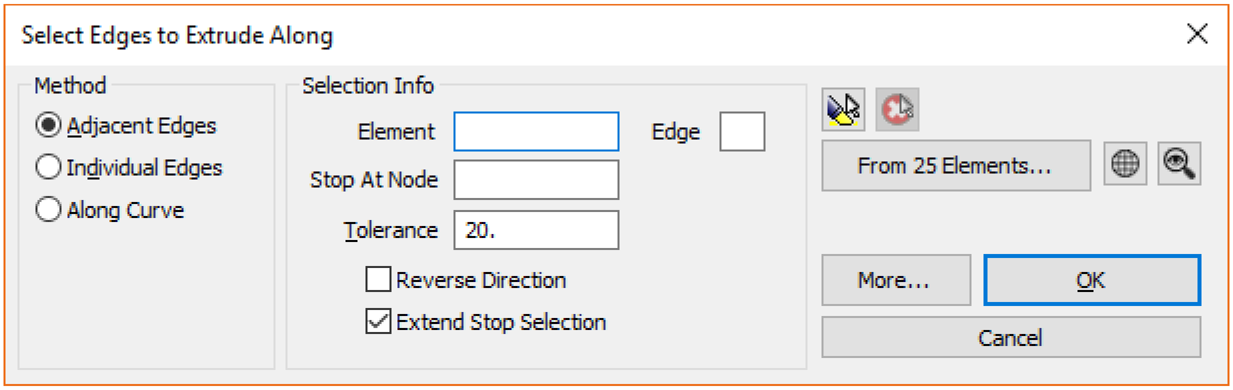
**7.** If using the *Along Curve* method, select the curve(s) to sweep along using the standard entity selection dialog box. If required, do NOT include the “alignment curve” in this selection. If the *Alignment Curve* option is enabled, FEMAP will now prompt for the “alignment curve”. If an alignment curve is not selected, FEMAP will then ask for a reference point. The elements will be oriented toward either the specified point or selected curve. For examples, see “Sweeping Along a Non-Planar Curve”.

**8.** If using the *Along Element Edges* method, the element edge(s) to sweep along will be selected using the *Select Edges to Extrude Along* dialog box, which is explained below in “Specifying Edge IDs”. This method requires a the use of a mapped mesh as the “sweep path”, as the nodes of the original elements being swept which are connected to the rest of the model must each have an element edge to follow for the full sweep path. If attempting to sweep along a non-mapped mesh, an error will be issued. For an example, see “Sweeping using Along Element Edges option”

***Note:*** The *Along Element Edges* method is designed to facilitate creation of a swept mesh for specific purposes, such as creating a solid mesh for a “weld” around pipes modeled with shell or solid elements.

### Specifying Edge IDs

The *Edge Selection* dialog box is used to select element edges to perform specific operations:




This dialog box provides several ways to select element edges. Simply choose the face graphically by moving the cursor near the center of the edge and clicking the left mouse button. The selected face will be highlighted. When using *Adjacent Edges*, and an unexpected edge is selected, simply move the mouse and click again until the desired edge is selected. When using *Individual Edges*, any unexpectedly chosen edge will need to be removed from the list of edges, otherwise it will remain selected.

It is possible to use a combination of different methods or the same method multiple times to select element edges. At any time, simply can click the *More...* button and the edges selected by the current *Method* will be added to the "overall edge selection list". A running count of edges currently in the "overall edge selection list" will appear in the upper right of the dialog box. If changing to a different *Method* while edges are selected by the current *Method*, but have not been added to the "overall edge selection list" via the *More...* button, FEMAP will ask "Ok to Select (#) Edges from Previous Method?" if a different *Method* is selected. If answering *Yes*, the faces will added to the "overall edge selection list", while answering *No* will disregard the faces selected with the most recent use of the previous *Method*, not all uses of the previous *Method*.

To limit which elements will be considered when using a method which automatically determines edges (for instance, *Adjacent Edges*), click the *From (#) Elements...* button to use the standard entity selection dialog box to select elements. Typically, it make sense to click the *Reset* button in the *Entity Selection* dialog box to clear the list of selected elements before choosing elements to limit edge selection. It is also possible to use the icon buttons to the right of the *From (#) Elements...* button to quickly *Select All* or *Select Visible* elements.

It may not always be obvious exactly which element edges will be selected by certain methods. To see a "preview" of selected element edges in the graphics window, click the *Show* icon button above the *From(#) Elements...* button and the element edges will be highlighted. When done with the "preview", simply click the *Show* icon button again.

Finally, all faces can be removed from the "overall face selection list" by clicking the *Clear List* icon button next to the *Show* icon button.

#### Choosing Adjacent Edges

The most powerful method for choosing edges, especially for complex solid and planar element models, is *Adjacent Edges*, which is the default. Simply choose just one initial edge (and the associated element ID). This can be done very easily by graphically selecting the edge. In order to select more or less edges, specify a tolerance angle. FEMAP will search all selected elements for edges that are connected to the chosen edge and that are within the specified tolerance from being colinear with an already selected edge. This can be used to find all edges on an outer edge of a solid or planar elements - regardless of the shape.

When selecting an edge graphically, the edge will highlight with an arrow head pointing in the "direction to search" for adjacent edges and only edges in that direction will be selected. The direction of the arrow depends on the proximity of the cursor to "Node A" or "Node B" on the edge. If the cursor is closer to "Node A", then the arrow will point from "Node B" to "Node A", while it will point from "Node A" to "Node B" when closer to "Node B". To switch the direction of an already selected edge, enable/disable the *Reverse Direction* option, as needed.

***Note:*** When graphically selecting an edge, the *Reverse Direction* option may become enabled. This depends on the order of the two nodes of the edge in the element specified in the *Element* field. To reverse the direction on edge of this type, simply disable to *Reverse Direction* option.

The *Stop At Node* option can be used to end selection of adjacent edges in the search direction at a particular node. When enabled, *Extend Stop Selection* allows the node specified for *Stop At Node* to not be a node on the selected edges. If this is the case, the node specified for *Stop At Node* is used as a "starting point" and extended out along the edges of connected elements, in all directions, to potentially limit selected edges in other sections of the model. Simply disable *Extend Stop Selection* to only have a node which is on adjacent edge(s) limit edge selection.

#### Choosing Individual Edges

This method simply allows you to choose individual element edges one at a time, by graphically selecting edges or by entering an *Element* ID and *Edge* ID, then clicking *Add to List* icon button. Edges can be removed from the list by highlighting any number of edges in the list, then clicking the *Remove from List* icon button. When an edge is highlighted in the list, the edge will also be highlighted in the graphics window.

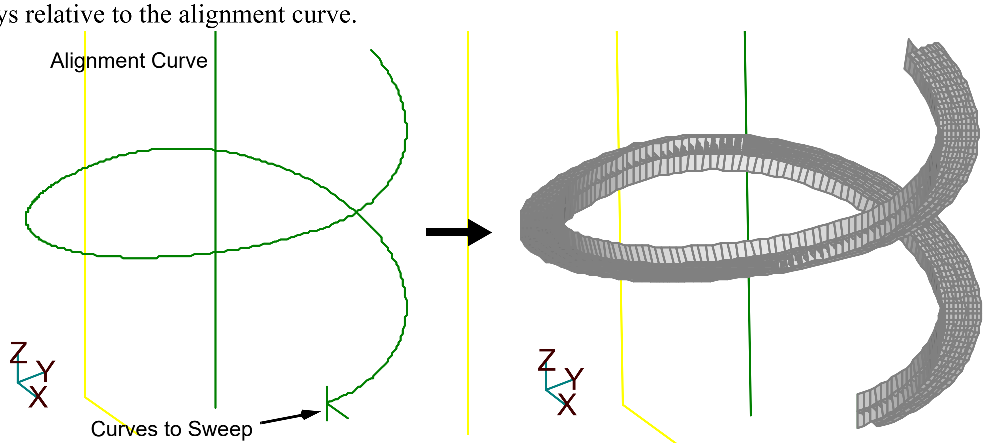
#### Choosing Edges Along a Curve

If the mesh is associated with a surface or solid, curves can be used to select element edges. Simply choose *Along Curve*, then select a curve and any element edges which contain nodes associated to the curve will be selected.

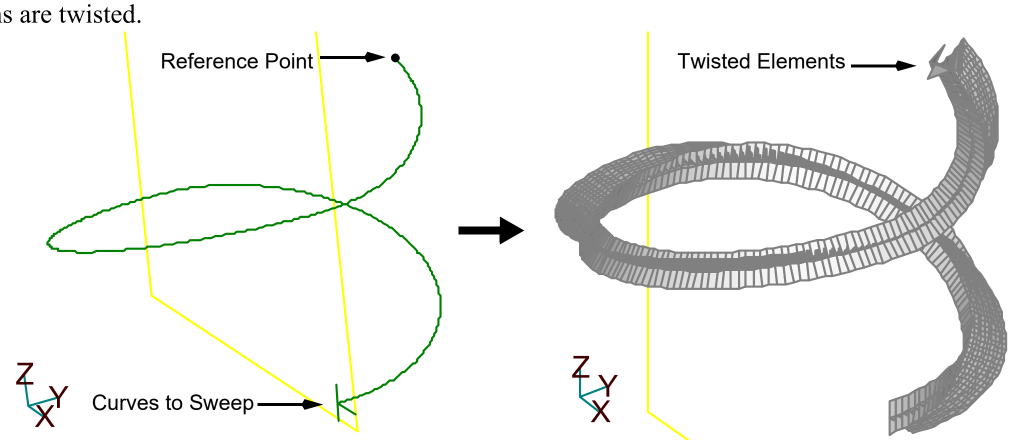
### *Sweeping Along a Non-Planar Curve*

If sweeping along a non-planar curve, either a reference point or alignment curve to orient the elements along the curve must be specified.

The figure shows how two curves are swept along an out of plane curve. Note how the orientation of the elements stays relative to the alignment curve.



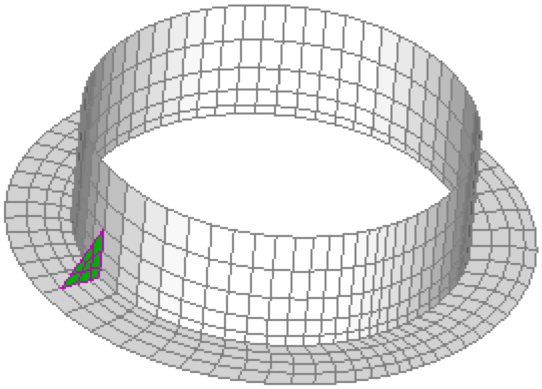

Sometimes you may prefer to use a reference point. However, poor placement of either an reference point or alignment curve can cause problems with the element orientation. In the example below, note how the element orientations are twisted.



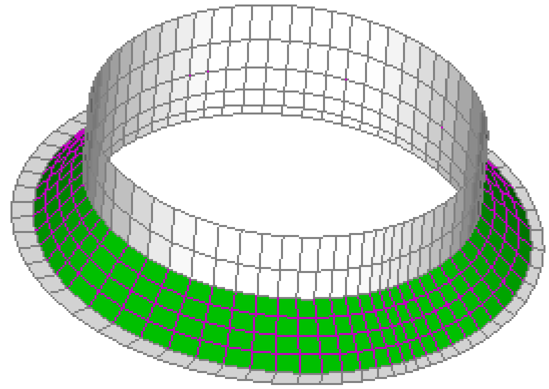

### *Sweeping using Along Element Edges option*

Here is an example of where using the *Along Element Edges* options can be beneficial. Although the mesh is biased in different directions, it is still a "mapped mesh". Sweeping the "green" elements along a curve could work, but the resulting elements would not "match" the existing mesh, therefore the nodes of the swept solid elements would not be coincident with the nodes of the shell elements, which may not be useful. Because the "green" elements have a mapped mesh to follow where they are connected to the "white" elements, this method can be used.

Green elements will be swept using White elements as a sweep path



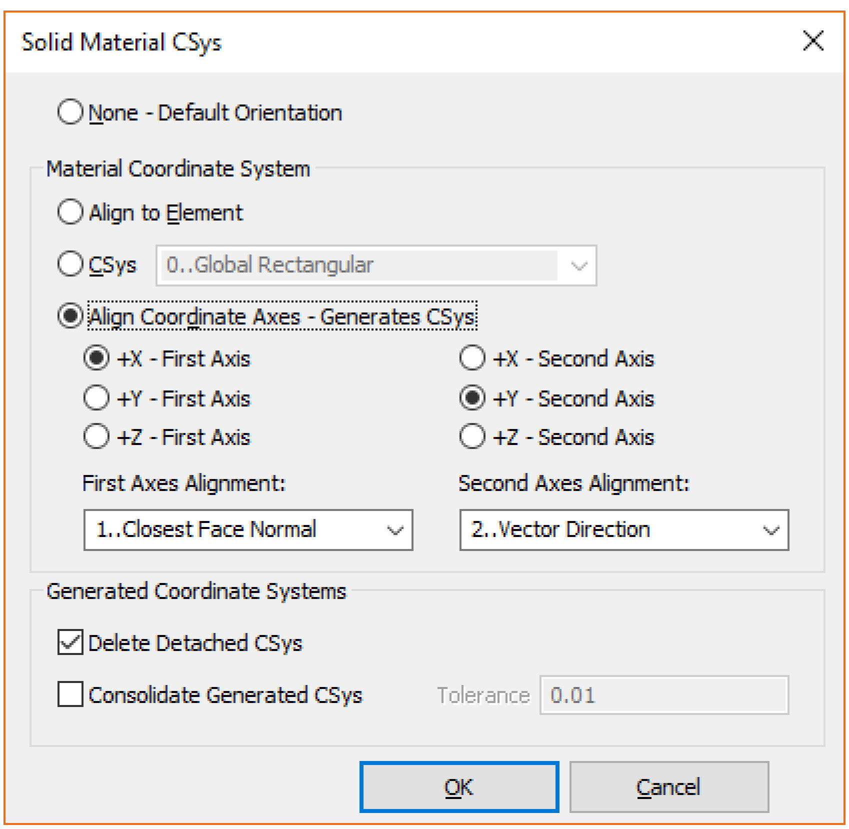
Notice how the green solid elements get larger and smaller as they “follow” the shell elements in the sweep path



## *Elements*

- Added options to the *Ansys Keyopt* section on *Ansys* tab of the Element Formulations dialog box to specify where results should be recovered for laminate and solid laminate elements. Sets the value of KEYOPT(8). For linear laminate elements, use the fourth drop-down in *Ansys Keyopt* section. For parabolic laminate elements set to “1..SHELL91” or “2..SHELL99”, use first drop-down or use the second drop-down when set to “0,,SHELL281”. For linear or parabolic solid laminate elements, use the third drop-down.
- Added *Modify, Update Elements, Solid Material CSys* command, which allows a material coordinate system override to be specified for solid, solid laminate, and/or solid cohesive elements.

Depending on the option selected in the *Material Coordinate System* section of the *Solid Material CSys* dialog box, an existing coordinate system, or any number of newly created coordinate systems generated by this command, will be assigned to the selected elements to use as a material coordinate system override. These overrides will be written out as MATCID entries to Simcenter Nastran.



There *Material Coordinate System* section contains three options:

**Align to Element** - sets the material coordinate system override for the selected elements to align to the element, which writes a MATCID entry using a value of -1 for the CID field, then the IDs of the selected elements.

**CSys** - uses the coordinate system selected via the drop-down as material coordinate system override for the selected elements, which writes a MATCID entry using the ID of the selected coordinate system for the CID field, then the IDs of the selected elements.

**Align Coordinate Axes - Generate CSys** - generates new coordinates systems, using a combination of the available options to specify two axes, to use as material coordinate system override for the selected elements. Writes a MATCID entry for each coordinate system generated by this command, which is also currently being used as a material coordinate system override, using the ID of the coordinate system for the CID field, then the IDs of the elements which reference the coordinate system.

For each axis, there are three options, *+X*, *+Y*, and *+Z*. The *First Axis* options are used to temporarily orient each newly created coordinate system based solely on the option selected via the *First Axes Alignment* drop-down, then the *Second Axis* option is used to complete orientation of the newly created coordinate systems based on the option selected via the *Second Axes Alignment* drop-down.

The *First Axes Alignment* and *Second Axes Alignment* drop-downs both contain six identical options:

"1..Closest Face Normal" - displays the *Select Element Faces to Align CSys Axis To* dialog box, which allows selection of element faces, then each element face's normal direction is used to align the selected axis.

"2..Vector Direction" - displays the *Vector Locate - Specify Vector to Align CSys To* dialog box, which is used to specify a vector to align the selected axis.

"3..Tangent to Curve" - displays the *Select Curve to Align CSys Axis To* dialog box, which is used to select a curve, then the location of the centroid of each selected element is projected onto the closest location on the curve and the tangent of the curve at that location is used to align the selected axis.

"4..Align to Surface u-Tangent" - displays the *Entity Selection - Select Surface(s) to Align CSys Axis To* dialog box, which is used to select any number of surfaces, then the location of the centroid of each element is projected onto the closest location on one of the selected surfaces and that surface's "u direction tangent" will be used to align the select axis.

"5..Align to Surface v-Tangent" - uses the same methodology as "4..Align to Surface u-Tangent", but aligns the selected axis to the "v direction tangent" of the appropriate surface for each element.

"6..Align to Surface Normal" - uses the same methodology as "4..Align to Surface u-Tangent", but aligns the selected axis to the normal direction of the appropriate surface for each element.

The *Generated Coordinate Systems* section contains options to potentially make the model less complicated:

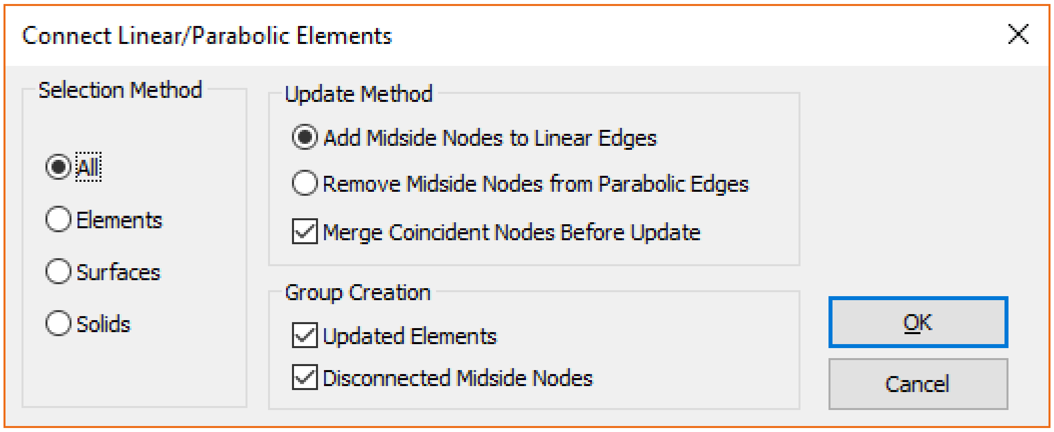
**Delete Detached CSys** - When enabled, which is the default, deletes any coordinate system which was being used as material coordinate system override for any of the selected elements before the command, but is no longer being used for that purpose after the command is completed.

***Note:*** If a coordinate system being used as a material coordinate system override is also referenced by any other entity in FEMAP, it will not be deleted.

**Consolidate Generated CSys and Tolerance** - When enabled, attempts to generate as few new coordinate systems as possible to properly assign material coordinate system overrides to all selected elements. *Tolerance* is an angular value and only coordinate systems where all three axes are within the tolerance will be consolidated.

- Updated *Modify, Project, Node* command by adding two options to the *Direction* section, *Radial Around Axis*, and *Radial Around Point*, which project nodes outwards from either a defined vector or specified location.
- Added *Modify, Update Elements, Connect Linear/Parabolic Elements* command, which updates the nodes on linear elements to match the nodes on parabolic elements which are connected to the linear elements or vice versa.

This command uses the *Connect Linear/Parabolic Elements* dialog box:




The *Selection Method* section is used to select which elements should be considered for update:

**All** - simply considers all elements in the model, therefore no additional selection is required

**Elements** - select the element(s) to consider for update using the standard entity selection dialog box

**Surfaces** - considers all elements associated to the selected surface(s) for update

**Solids** - considers all elements associated to the selected solid(s) for update

The *Update Method* section selects the method used to update the connected elements and sets an option.

When using *Add Midside Nodes to Linear Edges*, linear elements being updated will be changed to parabolic elements and midside nodes are added to any edge which is shared by a parabolic element.

When using *Remove Midside Nodes from Parabolic Edges*, parabolic elements being updated will have midside nodes removed from any edge which is shared by a linear element.

When *Merge Coincident Nodes Before Update* is enabled, which is the default, the nodes of the elements involved in the command, which depends on the option selected in the *Selection Method* section, will be merged using the model's "merge tolerance" before any elements are updated.

The *Group Creation* section offers two options to automatically create groups and both are enabled by default. When enabled, *Update Elements* will create a group containing all elements which were updated by the command. When enabled, *Disconnected Midside Nodes* will create a group of any midside nodes on parabolic elements which are connected to other elements at the corners, but not at all midside nodes.

## Properties

- Added the *ABAQUS Thermal...* button to the *Define Property - GAP Element Type* dialog box

For ABAQUS, gap properties are also used to define properties of interface elements, and you can specify the interface normal and width/area. The *ABAQUS Thermal...* button opens the *Define Gap Options* dialog box.

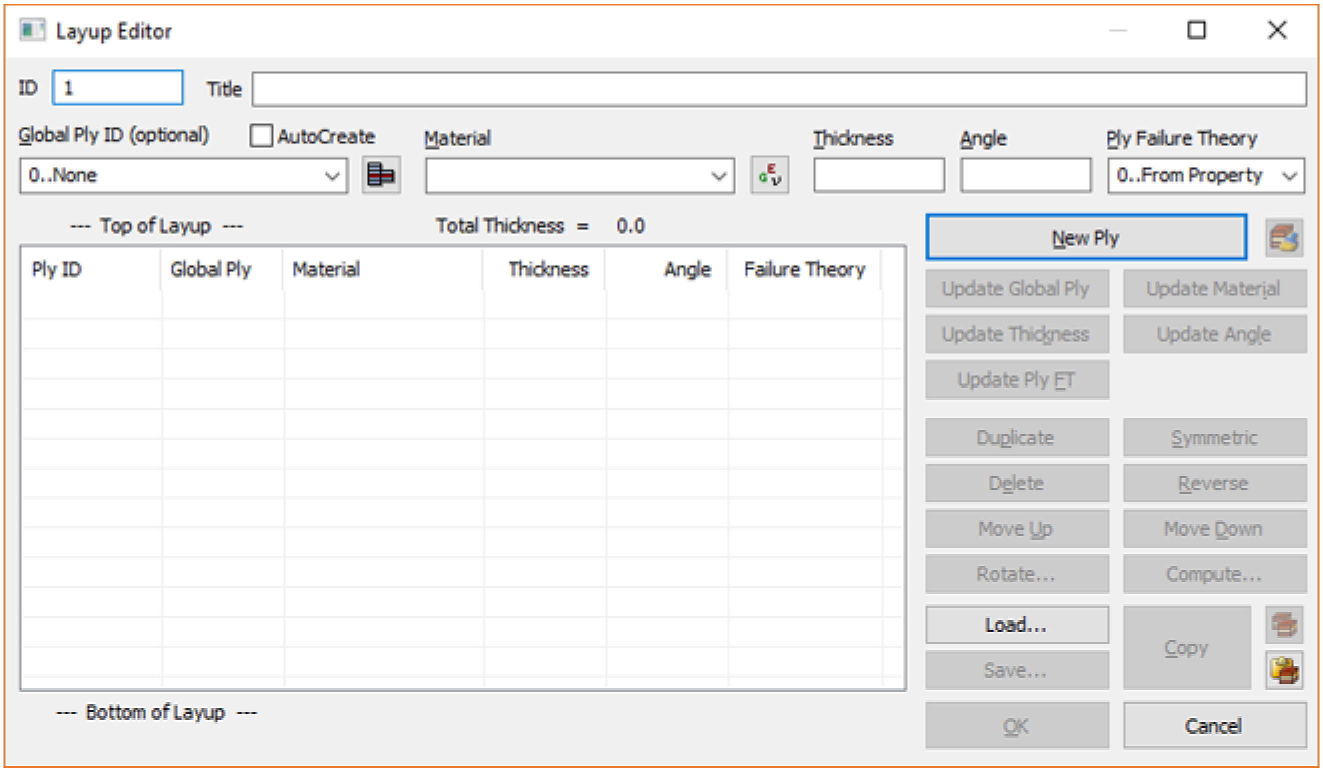
When *Active* is enabled in the *CONDUCTANCE* section, *Basis* is set to *Clearance*, and a function of type "15..Function vs. Value" is specified *vs. Clearance, Temperature*, a *GAP CONDUNCTANCE entry and list of *Conductance*, *Clearance*, *Temperature* (k[#], d[#], T[#]) entries are written, as needed, to define the functions in the ABAQUS input file. If *Basis* is set to *Pressure* and a function of type "15..Function vs. Value" is specified for *vs. Pressure, Temperature*, a *GAP CONDUNCTANCE, PRESSURE entry and list of *Conductance*, *Pressure*, *Temperature* (k[#], P[#], T[#]) entries are written, as needed, to define the functions in the ABAQUS input file.

When *Active* is enabled in the *RADIATION* section and a function is specified for *vs Gap Clearance*, a *GAP RADIATION entry, an entry containing *Emissivity A*, *Emissivity B* (Ea, Eb), and *Effective View Factor*, *Gap Clearance* (F[#], G[#]) entries are written, as needed, to define the functions in the ABAQUS input file.

## Layups

- Added ability to specify *Ply Failure Theory* ply-by-ply to support PCOMPG1 for SOL 401 and 402

The *Layup Editor* dialog box now contains an option to specify Ply Failure Theory ply-by-ply:

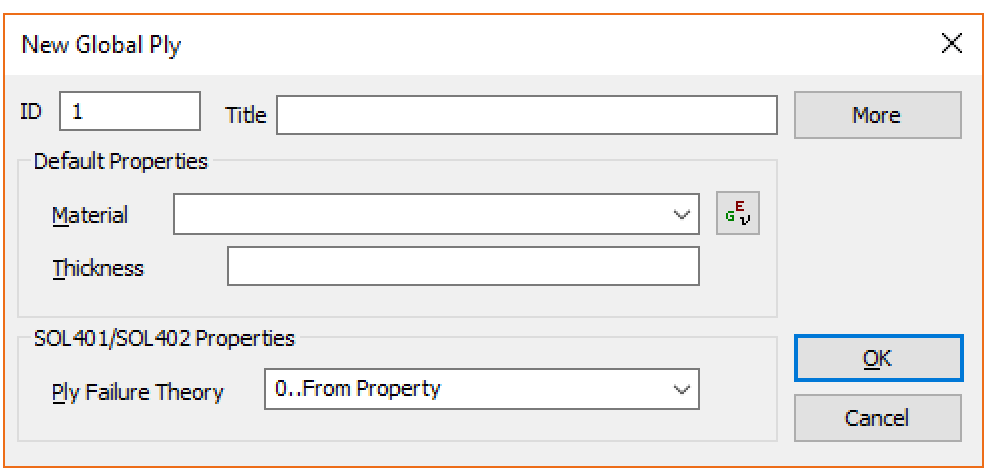



### Ply Failure Theory

The *Ply Failure Theory* drop-down allows selection of a failure theory for each ply. Available options are “0..From Property”, “1..Hill”, “2..Hoffman”, “3..Tsai-Wu”, and “4..Max Strain”. This option is currently only used by Simcenter Nastran SOL 401 and SOL 402 and will be ignored for all other solution sequences and by all other solvers. In addition, all plies must have a Global Ply assigned to write a PCOMPG1 entry to the Simcenter Nastran input file. The default value, “0..From Property”, indicates the ply will use the option specified in the *Failure Theory* section of the *Define Property - LAMINATE PLATE Element Type* dialog box which references the *Layup*.

- Added ability to specify *Ply Failure Theory* when creating or editing a *Global Ply.*

In the *Global Ply Definition* dialog box, create a new global ply by using the *New Ply* button. In the *New Global Ply* dialog box, enter a *Title* (up to 79 characters), and optionally choose a *Material*, enter a *Thickness*, and/or select a *Ply Failure Theory* (only used by Simcenter Nastran SOL 401 and 402).



## Loads and Constraints

- Added the ability to create *Bearing Force* and *Torque* loads on curves.

Two load types, *Bearing Force* and *Torque*, are available when applying loads to curves or surfaces. Both use the *Midside Node Adjustment* and *Total Load* options by default. These options are explained in greater detail below.

When creating a *Bearing Force*, enter a *Magnitude*, *Load Angle* (specifies the area in which the “bearing” is in contact, entered in degrees. 180 is the default), and *Phase* (if needed), then specify a vector to represent the direction of the load. Additionally, there is an option to have the Bearing Force be *Normal to Surface* (“on” by default), which will apply the loads radially to cylindrical surfaces. Also, the load may be a *Traction Load* (“off “by default), which will essentially have the load “pull” on a surface instead of “push”

When creating a *Torque* load, enter a *Magnitude* and *Phase* (if needed), then specify a vector representing the axis the load acts about.

#### Total Load

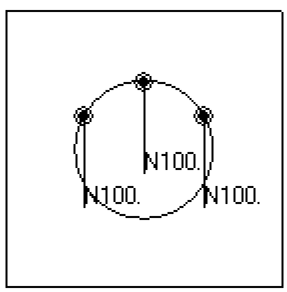
Only available for *Force*, *Torque*, *Bearing Force*, and *Moment*, load types applied to curves. Allows you to enter the “Total” Force, Torque, Bearing Load, or Moment to be applied over all selected curves, not applied to each curve. Uses curve length to spread the load out proportionally. *Total Load* is the default for these load types when more than one curve has been selected.

#### Normal to Surface/Traction Load

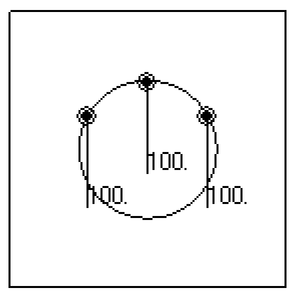
Only available when creating a *Bearing Force*. See “Bearing Load Example” above for more details.

### Bearing Load Example

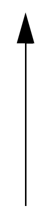
For example, 4 copies of “a square with a hole” geometry are all loaded with bearing forces using different options:



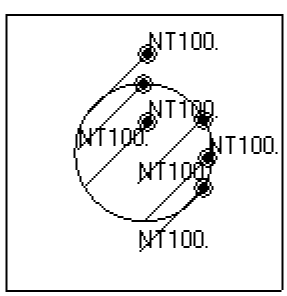

Bearing Force on single curve with Normal to Curve option On



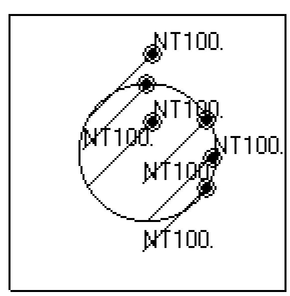

Bearing Force on single surface with Normal to Curve option Off



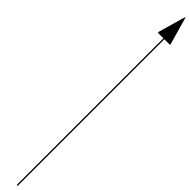
Vector used to define bearing force




Bearing Force on multiple surfaces with Total Load and Normal to Curve options On

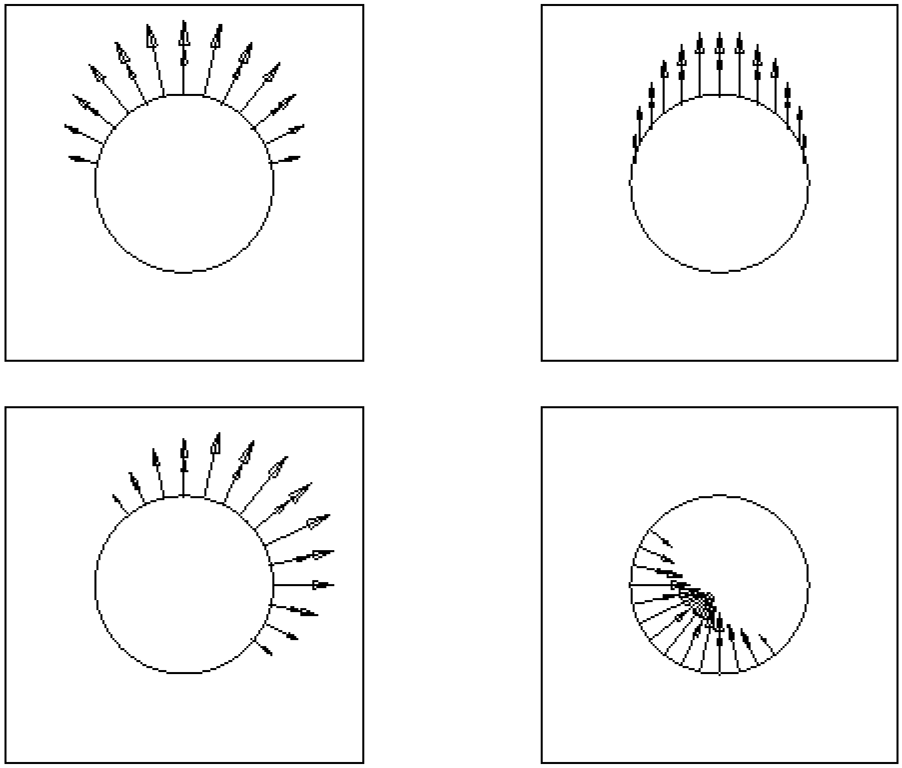



Bearing Force on multiple surfaces with Total Load, Normal to Curve, and Traction Load options On



Vector used to define bearing force

When expanded, Bearing forces will have varying values:



Bearing Forces from above shown Expanded
(Load values and mesh not shown for clarity)

## *Connections (Regions, Properties, and Connectors)*

- Updated the *Bolt Region* dialog box by adding the *Axes Determined by Solver* option to the *Solid Bolt Options* section. This option is only available when *Bolt Type* is set to *Solid*. When enabled, the *Bolt Axis CSys* and *Dir* controls become unavailable and can be helpful when creating certain types of bolts for Simcenter Nastran.
- Updated the titles of the tabs in the *Define Connection Property* dialog box used to specify connection property values for Simcenter Nastran (formally NX Nastran) to *Linear*, *Multistep Structural(401)*, *Multistep Kinematic (402)*, *Adv Nonlin (601)*, and *Explicit (701)*.
- Updated the look and feel of the *ANSYS* tab by isolating the *Friction Coefficient (MU)*, placing all the *Real Constants* in the upper portion of the dialog box, and all of the *Contact KEYOPTs* in the lower portion.
- Added *ABAQUS Thermal...* button to *ABAQUS* tab, which displays the *Define Gap Options* dialog box. See Properties section for a description of the options in the *Define Gap Options* dialog box.
- Added options to the *Multistep Structural (401)* tab, which are accessed by clicking the *More Options...* button. The new options are **Adaptively Modify Penalty Factor*, **Adjustment Tolerance*, **Const Offset Distance*, **End Time for Initial Penetrations*, and **Sliding Contact Formulation*.

***Adaptively Modify Penalty Factor** - Creates PENADAPT field in BCTPARM entry. Option to adaptively vary the contact stiffness between iterations.

When this box is not checked in FEMAP, no special field will be written to the BCTPARM and the penalty factor is not adaptively modified by the program. This is the default behavior.

When this box is checked, penalty factor is adaptively modified by the program.

***Adjustment Tolerance** - Creates ADJUST field in BCTPARM entry. Request for the software to remove initial separation by moving nodes. The software uses an adjustment tolerance to move the source nodes on to the target faces when the absolute value of the initial separation distance is below the adjustment tolerance. The adjustment tolerance is computed as *Adjustment Tolerance* value * characteristic length.

If *Adjustment Tolerance* = 0.0, only the nodes that are initially “over-closed” are adjusted as long as the over-closure is below the characteristic length.

If *Adjustment Tolerance* > 0.0, the adjustment is applied to nodes that are initially open with a gap smaller than value of *Adjustment Tolerance* value * characteristic length.

***Note:*** *Adjustment Tolerance* works from the node locations and *Initial Penetration* works from the Gauss point locations. If both are defined, *Adjustment Tolerance* is first applied, then *Initial Penetration*.

***Constant Offset Distance** - Creates OFFSET field in BCTPARM entry. Constant offset distance for contact pair.

This value overwrites the *Offset Distance* defined in the *Simcenter Nastran* section of the *Connection Region Options* dialog box accessed via the *Region Options...* button on the *Connection Region* dialog box. See "Region Options" for more information.

***Note:*** If both *Constant Offset Distance* and *Initial Penetration* are defined, the software adds the offset after it evaluates any gaps or penetrations as a result of the *Initial Penetration* setting.

***End Time for Initial Penetration** - Creates TZPENE field in BCTPARM entry. End time when initial penetrations will be eliminated. This value ramps the removal of initial penetrations.

For example, if a subcase start time is 5.0, end time is 10.0, and *End Time for Initial Penetration* =7.2, the initial penetrations will be eliminated starting from the beginning of the subcase, time = 5.0 until time = 7.2. Ramping the removal of initial penetrations can help convergence.

If *End Time for Initial Penetration* = 0.0, the initial penetrations are eliminated in the first subcase.

***Sliding Contact Formulation** - Creates the DISP field on the BCTPARM entry. Selects the small or large sliding contact formulation.

0..Automatically Update Contact Pairing - Contact pairing is automatically updated by the software based on the amount of relative sliding in the pair.

1..Update Contact Pairing Based on Initial Geometry - Small sliding formulation. Contact pairing is not updated with sliding displacements and is based on initial geometry.

2..Update Contact Pairing with Sliding Displacements - Large sliding formulation. Contact pairing is updated with sliding displacements.

***Note:*** When *Sliding Contact Formulation* is set to “1..Update Contact Pairing Based on Initial Geometry”, the *Geometry Updates* option in *Define Connection Property* (GUPDATE) is internally set to “0..None” (no pairing update).

When *Sliding Contact Formulation* is set to “2..Update Contact Pairing with Sliding Displacements”, *Geometry Updates* is internally set to “4..Each Iteration” (update every iteration).

When *Sliding Contact Formulation* is set to “0..Automatically Update Contact Pairing”, *Geometry Updates* is internally set to “3..Once per Step” for small displacement solutions (PARAM,LGDISP,-1) and “2..Start, SLIP>(GUPTOL*Avg Element Length)” for large displacement solutions ((PARAM,LGDISP,1)

## Optimization

- Added *Frequency Response - Nodal Displacement*, *Frequency Response SPC Forces*, and *Frequency Response Element Force, Stress, Strain* options to the *Category* drop-down in the *Optimization Response* dialog box used to define optimization limits. These options must be used to properly set optimization limits when *Analysis Type* is set to “4..Frequency/Harmonic Response” in the *NASTRAN Optimization Options* dialog box in the analysis set.
- Added Delete icon buttons to the Select Optimization Variable(s), Select Topology Region(s), Select Manufacturing Constraint(s), and Select Optimization Limit(s) dialog boxes in the Analysis Set Manager.

## Aeroelasticity

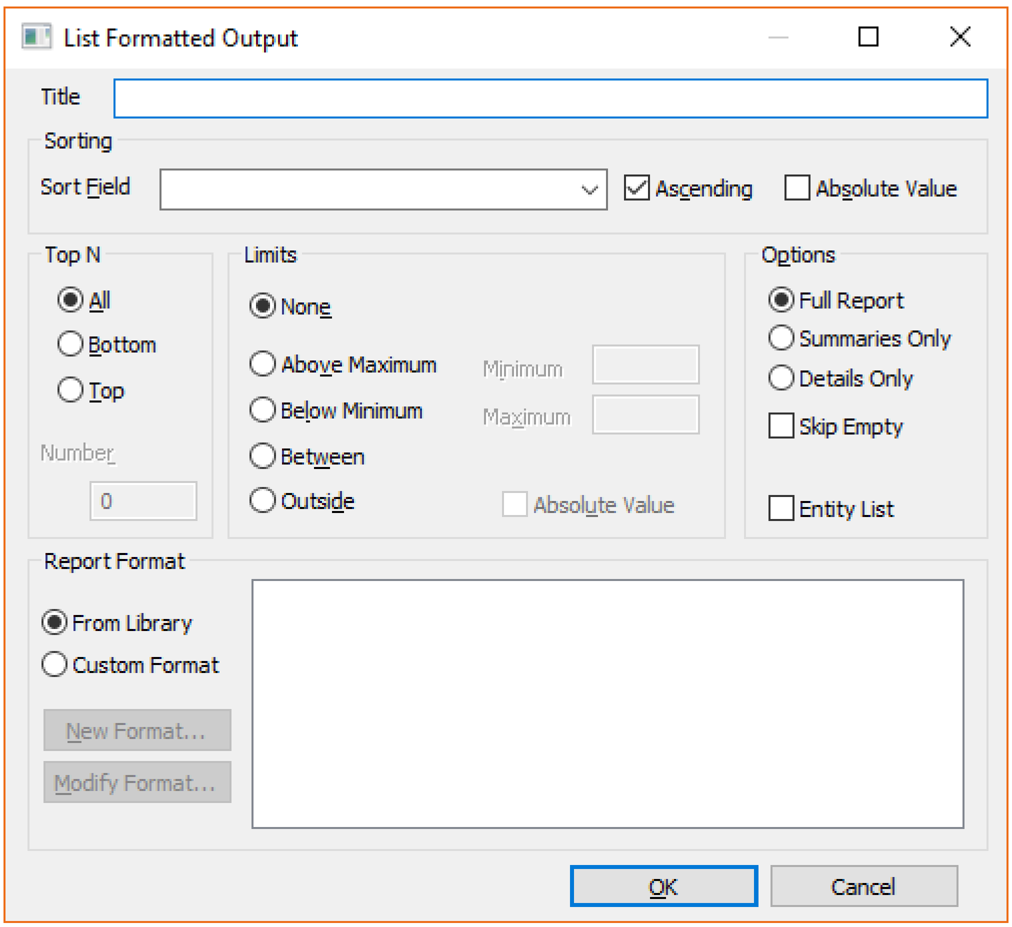
- Updated the display of aero control panels to show any control panel specified in the *Control Surface 2* section of the *Create Aero Control Surface* dialog box using a different shade of the color used for *Control Surface 1*.
- Updated the display of aero bodies to show lines along the length of the body instead of only at the slender body divisions. These lines are also drawn on any aero spline associated with an aero body.

## Listing

- Consolidated the functionality of the *List, Output, Standard* and *List, Output, Use Format* command into a single command, *List, Output, Formatted*, which lists output data using a format from a library or a custom format saved with the model.

To produce a formatted output report, first select the desired output set(s) using the *Select Output Set(s) to List* dialog box

The report format and other customization options are then specified using the *List Formatted Output* dialog box:




### Report Format

When using the *From Library* option in the *Report Format* section, formats which resemble those used by the various analysis programs are available via the *Femap Standard Libraries* tab of the *Select From Output Format Library* dialog box. Alternatively, use the *Personal* or *Shared* tab in the *Select From Output Format Library* dialog box to select formats which have been saved to a format library to create more custom listings and/or reports.

When *Report Format* is set to *Custom Format*, only formats saved with the model will be available. At any time, simply select a format from the list in the *Report Format* section.

### *Finishing the Report*

Once all of the desired options have been specified, click *OK*. If *Report Format* was set to *From Library*, select a format from a format library using the *Select From Output Format Library* dialog box.

Finally, whether the format was selected in the *List Formatted Output* dialog box or from the *Select From Output Format Library* dialog box, it is now time to select node(s) or element(s) using the standard entity selection dialog box. When selecting a format to list output for a particular element type, be sure to then select elements which have that type of output, otherwise, nothing will be listed. For example, if using the “NASTRAN CBUSH Forces” format, select only spring/damper elements which reference a spring/damper property with *Type* set to *CBUSH*.

- Updated the name of *List, Output, Format* command to *List, Output, Custom Format Definition* and moved it into a different section of the *List, Output* menu.

## Groups and Layers

- Added the *Group, Element, using Orientation Node* command, which will add any element which uses a selected node as an orientation node to the active group.

## Views

- Added ability to renumber *Views* using the *Modify, Renumber, View* command. Also, *Views* can now be renumbered using the *Modify, Renumber, All* command.
- Updated “Label Parameters” option in “Labels, Entities and Color” *Category* of *View, Options* command by replacing the *Label Font* box and the *Performance Graphics Font* button with two drop-downs in the *Font* section, which are used to specify the font and size to use for both entities supported by performance graphics and those that are not currently supported (i.e., all text in the view is shown using a single “unified font”).
- Updated “Performance Graphics” option in “Tool and View Style” *Category* of *View, Options* command by removing the *Performance Graphics Font* button, as all text in the view is shown using a single “unified font”.
- Updated “Contour/Criteria Style” option in “PostProcessing” *Category* of *View, Options* command by adding the “2..Contour with Zero” option to *Elements with no results*. Using “2..Contour with Zero” can aid in locating contact results which only exist on particular faces, but are obscured by other elements.
- Added “Region” option in “PostProcessing” *Category* of *View, Options* command, which controls if results on nodes or elements should be visible “through” any type of displayed region when a contour/criteria plot is also being displayed. In the case of *Connection Regions*, this can be quite useful for the results to be displayed “on top of” regions, especially when the analysis included any kind of contact. When enabled, which is the default, the *Contour/Criteria* option allows results to be visible “through” the displayed regions, while *Label Mode* controls if the output values will be visible through the displayed regions as well (“1..Allow Output Values”, default), or not (“0..No Labels”).

## Output and Post-Processing

- Added ability to import the deformations generated by a aeroelastic flutter analysis for the aero mesh on aero panels and aero bodies, then display those deformations using the various *Deformed Style* options available in the *View Select* dialog box or specified via *Style* drop-down in the *Deform* tool of the *PostProcessing Toolbox*.
- Added ability to display results “on top of” regions, which can be quite useful attempting to view results from an analysis included any kind of contact.

## Geometry Interfaces

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

| FEMAP Interface | Latest Supported Version |
|---|---|
| Parasolid | 31.1 |
| NX | 2019 |
| SolidWorks | 2019 |

- Updated STEP translator to use most recent version provided by geometry translator group at Siemens

For details, see “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 (Previously NX Nastran)
- Nastran Interfaces (NX and MSC/MD)
- MSC Nastran and Autodesk Nastran Interfaces
- ANSYS Interface
- 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 v2019.1 changes

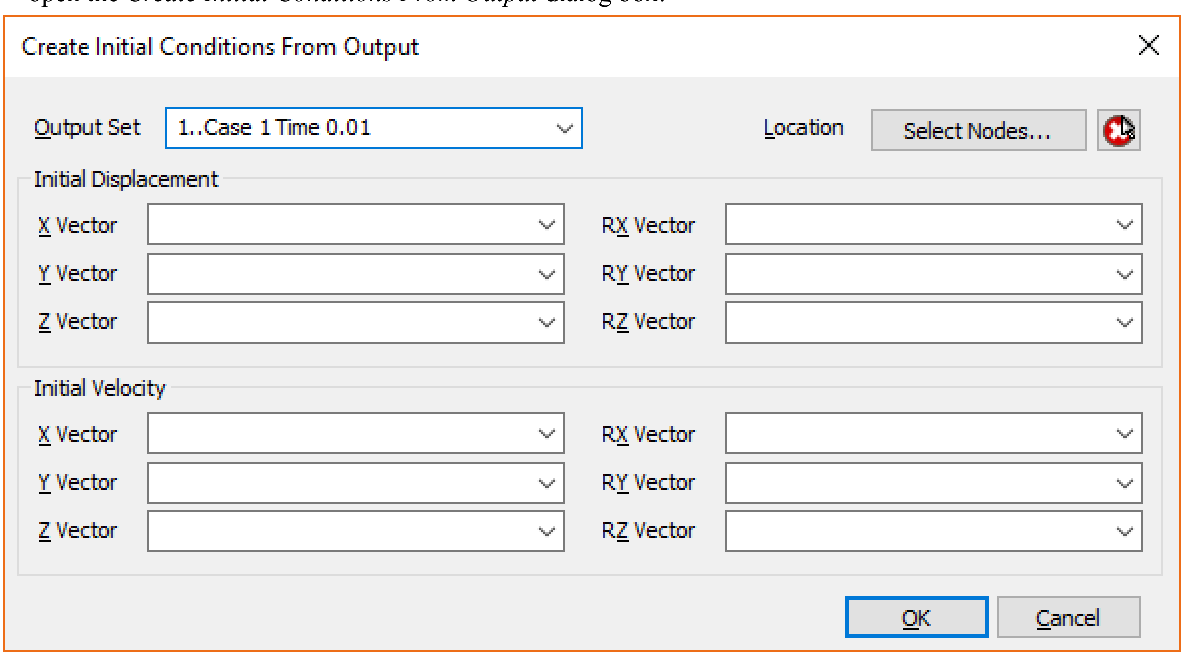
## Simcenter Nastran Interface (Previously NX Nastran)

- Updated NX Nastran to Simcenter Nastran throughout the FEMAP application.
- Added the ability to specify a material coordinate system override (MATCID entry) for solid, solid laminate, and solid cohesive elements, which is done via the *Modify, Update Elements, Solid Material CSys* command.
- Added support for glued contact in SOL 200.
- Added “4..Frequency/Harmonic Response” to the *Analysis Type* drop-down in the *NASTRAN Optimization Options* dialog box used to setup SOL 200 Topology Optimization. When defining optimization limits for “4..Frequency/Harmonic Response”, the *Frequency Response - Nodal Displacement*, *Frequency Response SPC Forces*, and/or *Frequency Response Element Force, Stress, Strain* options must be used.

### SOL 401 and SOL 402 Only

- Added ability to choose “3..Transient” for *Analysis Type* for the Master Case and Subcases
- Added *Create Initial Conditions From Output* icon button to *Boundary Conditions* dialog box

Only available when Analysis Type is set to “3..Dynamic” in the *Master Requests and Conditions* dialog box or the *Analysis Case* dialog box for any subcase. Click the icon button to the right of the *Initial Conditions* drop-down to open the *Create Initial Conditions From Output* dialog box:



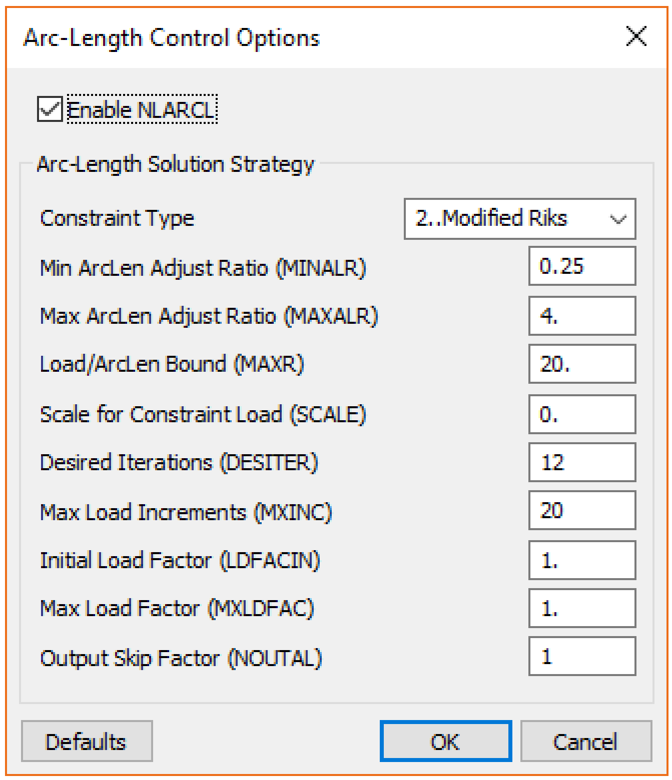
This dialog can be used to create a load set to specify *Initial Displacement* and/or *Initial Velocity* to use as *Initial Conditions*. Select an output set using the *Output Set* drop-down, then select any type of nodal output to use for enforced displacements (*X Vector*, *Y Vector*, and/or *Z Vector* in *Initial Displacement* section), enforced rotations (*RX Vector*, *RY Vector*, and/or *RZ Vector* in *Initial Displacement* section), velocities (*X Vector*, *Y Vector*, and/or *Z Vector* in *Initial Velocity* section), and/or rotational velocity (*RX Vector*, *RY Vector*, and/or *RZ Vector* in *Initial Velocity* section). Once the output has been selected, click the *Select Nodes...* button to use the standard entity selection dialog box to select nodes to create displacement and/or velocity loads.

### SOL 401 Only

- Added *Arc-Length Control Options* dialog box to specify parameters written to the NLARCL entry.

The Arc-Length Control methods allow for post-buckling analysis of structures with snap-through instability in Solution 401, which cannot be achieved through the Newton-Raphson solution strategy.

The *Arc-Length Control Options* dialog box contains a number of options which can be used to specify various parameters on the NLARCL entry for Simcenter Nastran Solution 401:




*Enable NLARCL* must be enabled before any of the options can be specified. The default value for each control option is specified when entering the dialog box for the first time. Once values have been changed, it is possible to return the default values using the *Defaults* button.

**Constraint Type** - Used to select the arc-length method. Writes TYPE field with selected option (Default = “2..Modified Riks”).

1..Riks - Selects the Riks arc-length method. Writes RIKS.

2..Modified Riks - Selects the modified Riks arc-length method. Writes MRIKS.

3..Crisfield - Selects the Crisfield arc-length method. Writes CRIS.

**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. Writes MINALR field and corresponding value.

**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. Writes MAXALR field and corresponding value.

**Load/ArcLen Bound (MAXR)** - Defines the overall upper and lower bounds on the load increment/arc-length in the subcase. Must be greater than 0.0, Default = 20.0. Writes MAXR field and corresponding value.

**Scale for Constraint Load (SCALE)** - Scale factor for controlling loading contribution in the arc-length constraint. Must be greater than or equal to 0.0, Default = 0.0. Writes SCALE field and corresponding value.

**Desired Iterations (DESITER)** - Desired number of iterations for convergence to be used for the adaptive arc-length adjustment. Must be an integer value greater than 0, Default = 12. Writes DESITER field and corresponding integer value.

**Max Load Increments (MXINC)** - Maximum number of controlled load increments done in the arc-length subcase. Must be an integer value greater than 0, Default = 20. Writes MXINC field and corresponding integer value.

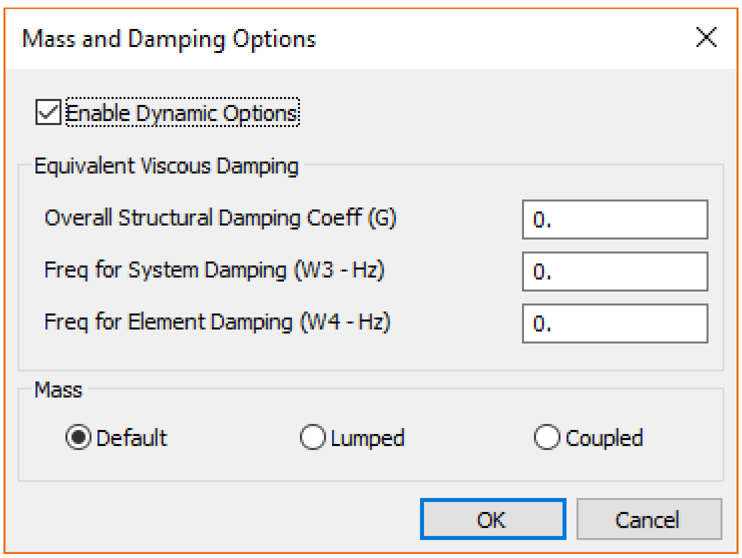
**Initial Load Factor (LDFACIN)** - Initial load factor. This load factor will be used to compute initial arc-length. Must be a value greater than 0.0, Default = 1.0. Writes LDFACIN field and corresponding value.

**Max Load Factor (MXLDFAC)** - Maximum value of load-factor at which solution will be terminated. Default = 1.0. Writes MXLDFAC field and corresponding value.

**Output Skip Factor (NOUTAL)** - Skip factor for output of the incremental results. Output always occurs at the final increment. For example, if NOUTAL=2 is defined, output occurs at every other converged solution increment and for the final increment. If NOUTAL=0 is defined, output only occurs at the final increment.

- Added *Mass and Damping* dialog box to specify mass and damping values for dynamic analysis.

The *Mass and Damping Options* dialog box is used to specify *Equivalent Viscous Damping* options, as well as an option to specify a *Mass* normalization method for dynamic analysis:




*Enable Dynamic Options* must be enabled before entering values and only used by the *Global Requests and Conditions* when *Analysis Type* is set to “3..Transient” or in any *Analysis Case* which is set to “3..Transient”.

There are three values which can be set in the *Equivalent Viscous Damping* section: *Overall Structural Damping Coeff (G)* creates the PARAM,G entry; *Freq for System Damping (W3 - Hz)* creates the PARAM,W3 entry; and *Freq for Element Damping (W4 - Hz)* creates the PARAM,W4 entry.

The *Mass* section designates if mass matrices for elements with coupled mass capability (i.e., CBAR, CBEAM, CHEXA, CPENTA, CPYRAM, CQUAD4, CQUAD8, CQUADR, CQUADX4, CQUADX8, CROD, CTETRA, CTRAX3, CTRAX6, CTRIA3, CTRIA6, CTRIAR, and CTUBE.) should be “Coupled” or remain “Lumped” (default for most analyses) by writing the PARAM, COUPMASS entry in the Nastran input file. When “Coupled” is selected, both structural and non-structural mass are taken into account for the aforementioned elements.

- Added *Time Integration* section to *Solution and Convergence Options* dialog box.

**Integration Scheme (TINTMTH)** - Integration scheme. Writes TINTMTH with the selected option (Default = “1..Newmark”). Choose from:

“1..Newmark” implicit predictor-corrector. Writes NEWMARK.

“2..Hilber-Hughes-Taylor” (HHT) implicit predictor-corrector scheme. Writes HHT.

“3..Generalized Alpha” (with Theta). Writes GENALP.

“4..Modified Generalized Alpha” (with Theta). Writes MGENALP.

**Param1 (BETA)** - Newmark scheme parameter 1, Beta. Must be greater than 0.0, Default = 0.25. Writes BETA field and corresponding value.

**Param2 (GAMA)** - Newmark scheme parameter 2, Gamma. Must be greater than 0.0, Default = 0.5. Writes GAMA field and corresponding value.

**HHT Scheme Param (ALFA)** - HHT scheme parameter, alpha. Must be between 0.0 and 0.333333, Default = 0.05. Writes ALFA field and corresponding value.

**Generalized Alpha Methods Parameter (TETA)** - Generalized alpha methods parameter, Theta. Writes TETA field and corresponding value. Default = -0.97 for “3..Generalized Alpha” or 0.33 for “4..Modified Generalized Alpha”.

### SOL 402 Only

- Added ability to choose “7..Buckling” for *Analysis Type* for Subcases. When used, adds a *Modal* item to tree for the subcase, which opens the *NASTRAN Model Analysis* dialog box.
- Added *Viscous Material Option* section of the *Multi-Step Control Options* dialog box.

**Integration Error Control (TSVSC)** - Integration error control. When disabled, which is the default, writes nothing. When enabled, writes TSVSC with corresponding characters, ON.

**Error Threshold (VSCOTE)** - Viscous material integration error threshold. Must be greater than or equal to 0.0, Default = 0.1. Writes VSCOTE field and corresponding value.

**Min Stress Factor (VSCOSN)** - Viscous material minimum stress normalization factor. Must be an integer value above 0, Default = 1. Writes VSCOSN field and corresponding integer value.

- Added options to the *Time Step* section of the *Multi-Step Control Options* dialog box. Also, updated many of the names of the other options in the section to match what is now expected by Simcenter Nastran.

**Number of Iterations (ITEREF)** - Number of iterations. Must be an integer value above 0, Default = 0.6 * *Max Iter (ITMA)* specified in the *Solution and Convergence Options* dialog box. Writes ITEREF field and corresponding integer value.

**Integration Error Control (TSDYN)** - Integration error control. When disabled, which is the default, writes nothing. When enabled, writes TSDYN with corresponding characters, ON.

**Zero Pivot Time Step Rejection (RJPZ)** - Zero pivot time step rejection. When disabled, which is the default, writes nothing. When enabled, writes RJPZ with corresponding characters, ON.

**Negative Pivot Time Step Rejection (RJPN)** - Negative pivot time step rejection. Writes RFPN with the selected option (Default = “0..Never”).

“0..Never” - Writes nothing.

“1..Last Iteration Only” - Writes a value of 1.

“2..Print State at Last Iteration” - Writes a value of 2.

“3..Always” - Writes a value of 3.

- Added options to the *Analysis Control* section of the *Multi-Step Control Options* dialog box.

**Unsymmetrical Matrices for Complex Modes (MATSYM)** - Unsymmetrical stiffness, damping, and mass matrices for complex modes. When enabled, which is the default, writes nothing is written. When disabled, writes MATSYM entry with corresponding characters, NO.

**Enable Inertia in Dynamics (INERTIA)** - Take inertia into account in nonlinear dynamic subcases. When enabled, which is default, writes nothing. When disabled, writes INERTIA with corresponding characters, NO.

- Added option to the *Other Options* section of the *Multi-Step Control Options* dialog box.

**Creep Integration Factor (CRINFAC)** - Integration factor used to calculate incremental creep strain. Must be value from 0.0 to 1.0. Default is 1.0. Writes CRINFAC entry with corresponding value.

- Updated options available for *Integration Scheme (TINTMTH)* in the *Time Integration* section of the *Solution and Convergence Options* dialog box.

**Integration Scheme (TINTMTH)** - Integration scheme. Writes TINTMTH with the selected option (Default = “4..Generalized Alpha”).

“1..Newmark” implicit predictor-corrector. Writes NEWMARK.

“2..Hilber-Hughes-Taylor” (HHT) implicit predictor-corrector scheme. Writes HHT.

“3..Generalized Midpoint” (with theta). Writes GENMID

“4..Generalized Alpha” (with theta). Writes MGENALP.

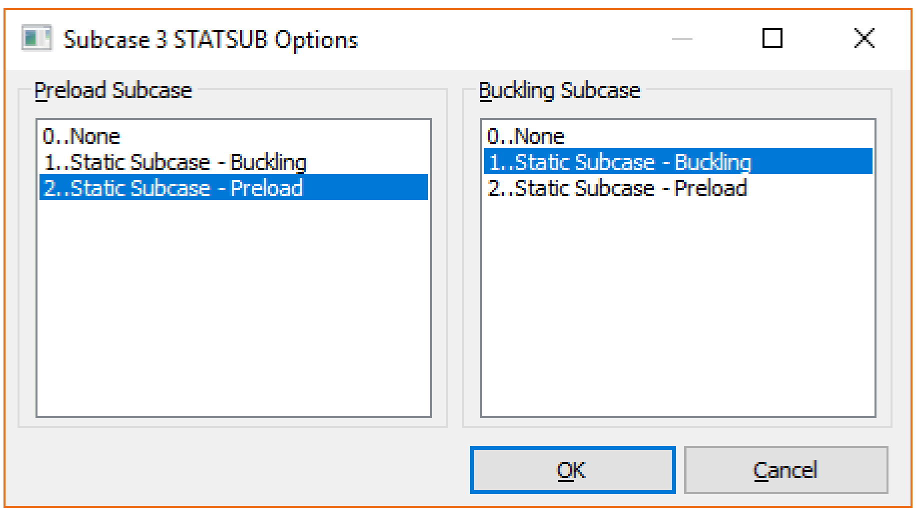
A number of bugs were corrected

For details, see Section 8, “Analysis Program Interfaces” in the *FEMAP User Guide*.

### *Nastran Interfaces (NX and MSC/MD)*

- Added ability to import Aero Mesh deformations from Flutter Analysis from .f06 file (only noticeable difference to the user is that new output vectors are available).
- Added *STATSUB Options* dialog box for buckling subcases.

If performing buckling analysis, and two subcases exist, the *STATSUB Options* dialog box can be used to select specific static subcases, which are then referenced by a STATSUB (BUCKLING) and, optionally, a STATSUB (PRELOAD) written to the case control section of a particular buckling subcase.




***Note:*** To access the STATSUB options for a particular subcase in FEMAP, “expand” the *Case* portion of the tree of a a subcase in a Buckling Analysis Set in the *Analysis Set Manager*, highlight *STATSUB* from the list, then click the *Edit* button. The *Subcase # STATSUB Options* dialog box will appear.

This dialog box can also be used to explicitly choose which static subcase should be used with each buckling subcase when setting up a multi-case buckling analysis. In addition, can be beneficial when attempting to “round tripping” a Nastran input files for buckling analysis, which was either A) not created using FEMAP or B) contain multiple subcases which contain a STATBUB entry referencing the same static subcase.

- Added more generalized read support for THRU fields for SPC1, ASET1, BSET1, etc entries, as long as the form “a” THRU “b” is used and all three fields reside on a single line, as it cannot continue across lines.
- Added *Contact* section to *NASTRAN Output Requests* dialog box and added options for *Contact* and *Glue*.
- Added “4..Frequency/Harmonic Response” to the *Analysis Type* drop-down in the *NASTRAN Optimization Options* dialog box used to setup SOL 200 Design Optimization. When defining optimization limits for “4..Frequency/Harmonic Response”, the *Frequency Response - Nodal Displacement*, *Frequency Response SPC Forces*, and/or *Frequency Response Element Force, Stress, Strain* options must be used.

- Added *Buckling* option to the *NASTRAN Modal Analysis* dialog box which can be accessed in an analysis set which has *Analysis Type* set to "10..Nonlinear Static". When *Buckling* is enabled, the *Inverse Power/Strum* method will be selected automatically and some additional options are available. Writes PARAM,BUCKLE,2, which requests buckling in a SOL 106 cold start run, is written along with an EIGB entry.

A number of bugs were corrected

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

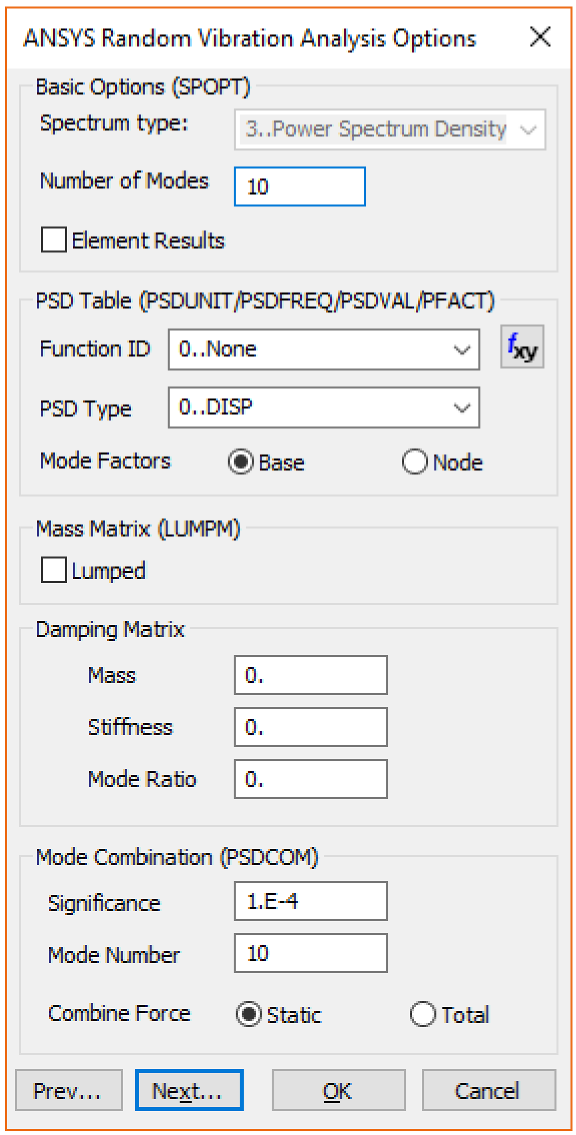
## MSC Nastran and Autodesk Nastran Interfaces

- Added support to read and write pyramid elements

## ANSYS Interface

- Added ability to set up a random response analysis by selecting "6..Random Response" from the *Analysis Type* drop-down in the *Analysis Set* dialog box. Options for random response are specified in the *ANSYS Modal Analysis Options* dialog box and the new *ANSYS Random Vibration Analysis Options* dialog box.

The *ANSYS Random Analysis Options* dialog box is used to specify options for frequency response analysis:




### Basic Options (SPOPT)

The options in this section specify a the Spectrum Type and other overall options for random response or spectrum analysis. Writes the SPOPT entry with appropriate names, values, etc for the current case. For detailed information, see ANSYS documentation for SPOPT.

**Spectrum Type** - Specifies the type of spectrum for the analysis. Currently, the only available option is “3..Power Spectral Density”, but other options exist, which may be supported in a future release.

“0..Single-point Response Spectrum” - Writes SPRS to “Sptype” field on SPOPT

“1..Multi-point Response Spectrum” - Writes MPRS to “Sptype” field on SPOPT

“2..Dynamic Design Analysis Method” - Writes DDAM to “Sptype” field on SPOPT

“3..Power Spectral Density” - Writes PSD to “Sptype” field on SPOPT

**Number of Modes** - Specifies to use only the first “number of modes” from the modal analysis (i.e., value of “5” means to only use the first 5 modes from the modal analysis). Writes integer value to NMODE field on SPOPT. Maximum value allowed is 10,000.

**Element Results** - Element Results Calculation Key. When disabled, which is the default, writes NO to Elcalc field on SPOPT. When enabled, writes YES to Ecalc field.

### PSD Table (PSDUNIT/PSDFREQ/PSDVAL/PFACT)

The options in this section specify various options for the PSD function. For detailed information, see ANSYS documentation for PSDUNIT, PSDFREQ, PSDVAL and PFACT.

**Function ID** - Select an existing FEMAP function (“3..vs. Frequency”) to specify the power spectral density (PSD) values and corresponding frequency values for random response analysis. Alternatively, use the *New Function* icon button to create a new function containing PSD values and corresponding frequency values. The PSD values in the function are written to the PSDVAL entry, while the frequency values are written to the PSDFREQ entry.

**PSD Type** - Used to specify the type and units of the power spectral density (PSD) input. Writes PSDUNIT entry.

“0..DISP” - Displacement Spectrum (displacement$^2$/Hz). Writes DISP to “LABEL” field on PSDUNIT entry.

“1..VELO” - Velocity Spectrum (velocity$^2$/Hz). Writes VELO to “LABEL” field on PSDUNIT entry.

“2..ACEL” - Acceleration Spectrum (acceleration$^2$/Hz). Writes ACEL to “LABEL” field on PSDUNIT entry.

“3..ACCG” - Acceleration Spectrum (g$^2$/Hz). Writes ACCG to “LABEL” field on PSDUNIT entry.

“4..FORC” - Force Spectrum (force$^2$/Hz). Writes FORC to “LABEL” field on PSDUNIT entry.

“5..PRES” - Pressure Spectrum (pressure$^2$/Hz). Writes PRES to “LABEL” field on PSDUNIT entry.

**Mode Factors** - Calculates participation factors for the PSD input based on excitation location. When set to *Base*, specifies base excitation and writes BASE to Excit field of PFACT entry, while *Node* specifies nodal excitation and writes NODE to Excite field.

### Mass Matrix (LUMPM)

When the *Lumped* option is disabled, which is the default, nothing is written and the analysis uses the element-dependent mass matrix formulation. When *Lumped* is enabled, writes LUMPM,ON and the analysis uses a lumped mass approximation.

### Damping Matrix

The values can be used to specify damping options for random response analysis.

**Mass** - Specifies Mass Matrix multiplier, Alpha, for damping to form the viscous damping matrix [C]. Writes the ALPHAD entry with value.

**Stiffness** - Specifies Stiffness Matrix multiplier, Beta, for damping to form the viscous damping matrix [C]. Writes the BETAD entry with value.

**Mode Ratio** - Defines constant modal damping ratio. Writes DMPRAT entry with value.

### Mode Combination (PSDCOM)

The values are to specify the power spectral density (PSD) mode combination method and additional values and writes PSDCOM entry.

**Significance** - Specifies a threshold value to be considered for mode combination. Only modes which have a significance level above this threshold, which is determined by the modal covariance matrix term divided by the maximum modal covariance term, will be combined. If a mode has a significance level below this threshold, it is considered insignificant and not contributing the modal combination. The higher the value, the less modes will be used. If set to 0.0, all modes will be combined. Writes the value to the SIGNIF field on PSDCOM entry.

**Mode Number** - Specifies to use only the first "number of modes" to be combined (i.e., value of "5" means to only use the first 5 modes for mode combination). Value must always be less than or equal to *Number of Modes* value in *Basic Options (SPOPT)* section. Writes integer value to COMODE field on PSDCOM.

**Combine Force** - Identifies the forces to be combined. When set to *Static*, which is the default, combines only the model static forces (writes STATIC to ForceType field of PSDCOM entry), while *Total* combines both the modal static and inertial forces (writes TOTAL to ForceType field of PSDCOM entry).

- Added *Mass Matrix (LUMPM)* and *Damping Matrix* sections to the *ANSYS Modal Analysis Options* and *ANSYS Harmonic Analysis Options* dialog boxes.

#### Mass Matrix (LUMPM)

When the *Lumped* option is disabled, which is the default, nothing is written and the analysis uses the element-dependent mass matrix formulation. When *Lumped* is enabled, writes LUMPM,ON and the analysis uses a lumped mass approximation.

#### Damping Matrix

The values can be used to specify damping options for random response analysis.

**Mass** - Specifies Mass Matrix multiplier, Alpha, for damping to form the viscous damping matrix [C]. Writes the ALPHAD entry with value.

**Stiffness** - Specifies Stiffness Matrix multiplier, Beta, for damping to form the viscous damping matrix [C]. Writes the BETAD entry with value.

**Mode Ratio** - Defines constant modal damping ratio. Writes DMPRAT entry with value.

- Added support to read and write bolt regions and bolt preloads for ANSYS. Bolt Regions create PSMESH and PRETS179 entries, while Bolt Preloads create SLOAD entries.
- Added support to read and write non-normal pressure loads on 2D/3D elements edges/faces. Distributed non-normal pressures are mapped to ANSYS input files as SFE command together with SURF153, SURF154 or SURF156 elements.
- Added *Skip Beam/Bar Cross Sections* option to *ANSYS Command and Model Control* dialog box. When enabled, writes all beam and bar properties to the ANSYS input file as SECTYPE, #, BEAM, ASEC, along with the corresponding computed property values from the *Define Property - BEAM Element Type* dialog box as SECDATA, regardless of how the beams were defined.
- Added support to read and write KEYOPT(8) for laminate and solid laminate elements. This value is stored as a formulation specify this value in FEMAP, use the Modify, Update Elements, Formulation command.

A number of bugs were corrected.

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

### *ABAQUS Interface*

- Added support to read and write *GAP CONDUNCTANCE, *GAP CONDUNCTANCE, PRESSURE, and/or *GAP RADIATION entries. To create or edit these values in FEMAP, use the *ABAQUS Thermal...* button in the *Define Property - GAP Element Type* dialog box or the on the *ABAQUS* tab of the *Define Connection Property* to open the *Define Gap Options* dialog box. For more information, see the Properties section.

A number of bugs were corrected.

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

## *Tools*

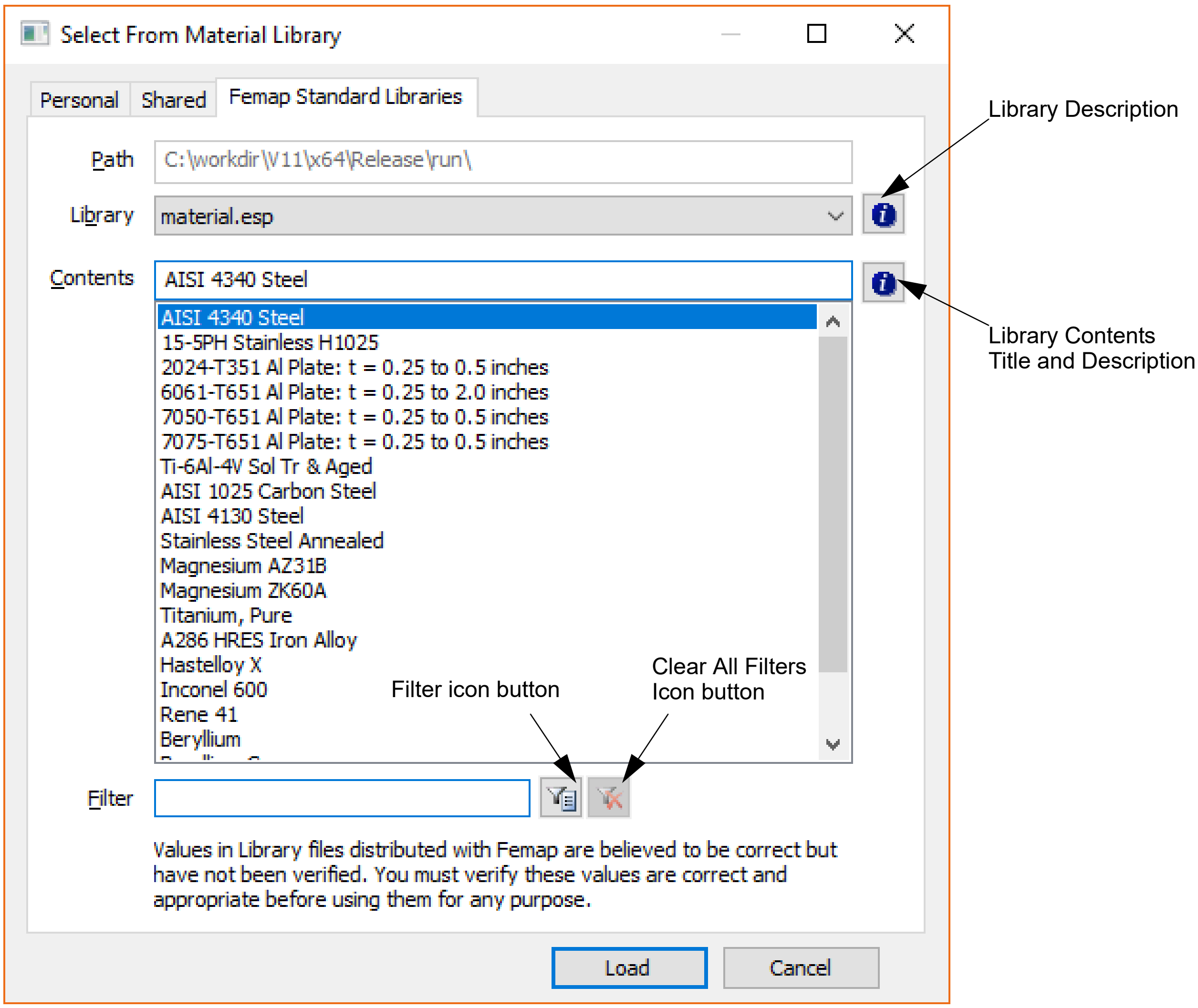
- Added the *Tools, Structural Analysis Toolkit* command, which simply opens the Structural Analysis Toolkit. This command is only available when the Structural Analysis Toolkit is installed and properly licensed.
- Updated the *Tools, Parameters* command by adding *Mesh Point* to specify *Color*, *Next ID*, and *Inc* for mesh points. In addition, added *Layup* and *Function* to specify Next ID and Inc for layups and functions, respectively.

- Updated the *Tools, Text* command by replacing the single *Font* drop-down, which offered only 77 combinations for font and, with two drop-downs, one to select the *Font* and the other to specify the size of the font.

## Libraries

- Updated libraries in a number ways, including a new dialog box with various tabs and controls which offer expanded functionality, storing of "personal" libraries in the user directory instead of the FEMAP installation directory, and a new preference to specify a single location for libraries "shared" with other users.

When selecting materials, properties, views, or other entities from a FEMAP library, the *Select From Library* dialog box will be displayed:




Depending on which preferences have been specified on the *Library/Startup* tab of the *Preferences* dialog box, the *Select From Library* dialog box will contain two tabs (*Personal* and *Femap Standard Libraries*) or three tabs (*Startup Shared Libraries* preference potentially adds a *Shared* tab).

All tabs share some functionality, including:

The *Path* field, which displays the directory path to available libraries for a particular tab

Ability to change the currently library for a tab, on-the-fly, via the *Library* drop-down

A *Contents* list, which contains the items available from current library

Ability to reduce the number of items in the *Contents* list via the *Filter* options

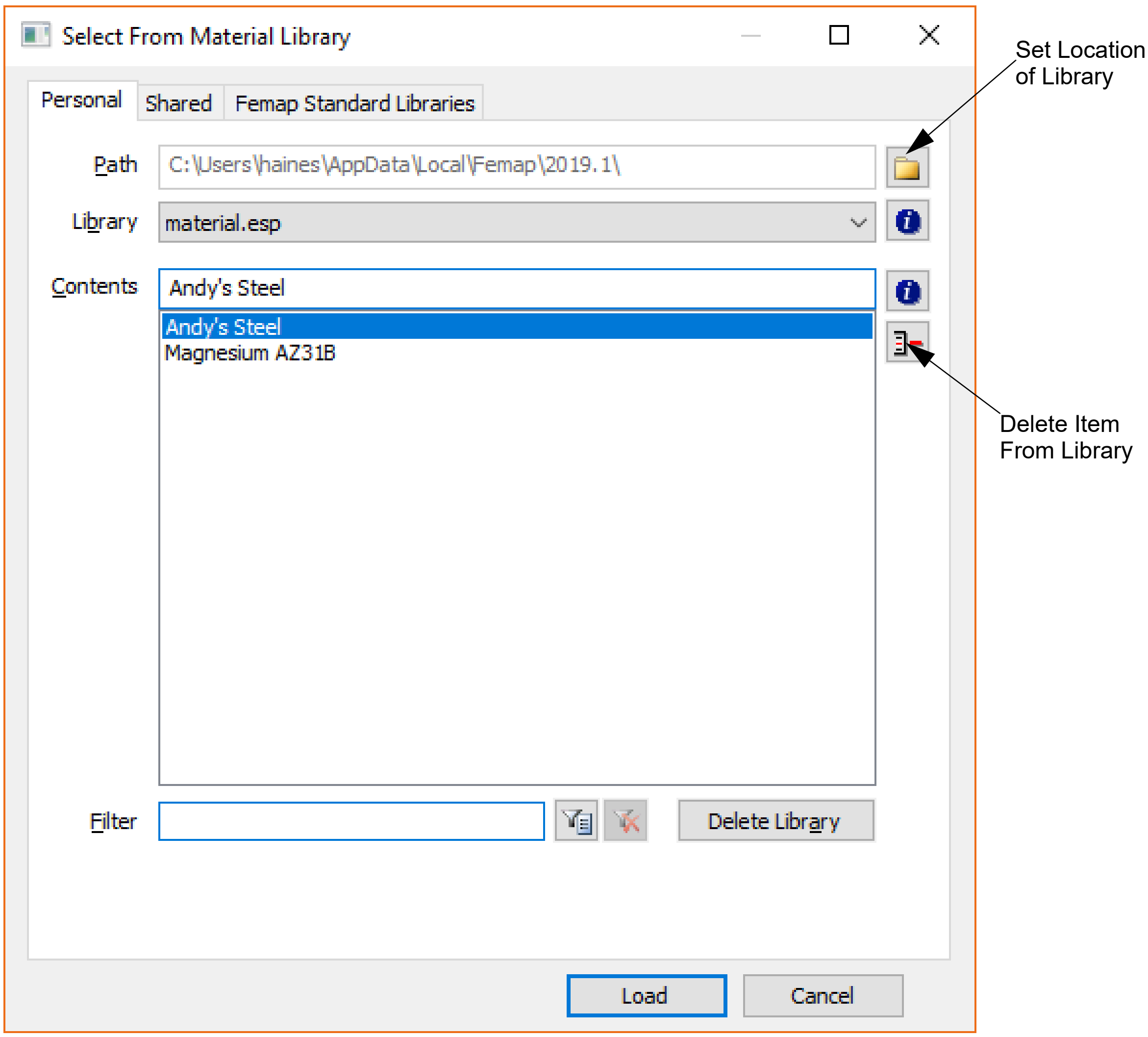
Icon buttons which can be used to show an overall description of the selected library (*Library Description*) or a description of a particular entity in the library (*Library Contents Title and Description*).

Each tab also offers some unique functionality, which is described here in greater detail.

### Personal

The *Personal* tab offers the ability to load an item from a library file (*.esp file) which is maintained by the user. This indicates items can be saved to a library, at any time, via the *Save* button in the creation dialog boxes for a specific entity type, or deleted from a library using the *Delete from Library* icon button to the right of the *Contents* list.

The *Set Location of Library* icon button can be used to select a different directory path, using the standard File selection dialog box, to find libraries for a specific entity type. The selected path will persist until the FEMAP application is closed completely. The *Delete Library* button deletes the entire library file which is currently specified by the *Library* drop-down.




Any entity which is saved when the *Personal* tab is active will be saved to the library file currently specified via the *Library* drop-down. In addition, the default library on the *Personal* tab for each entity type, at least when FEMAP is initially started, can be specified using the *File, Preferences* command, choosing the *Library/Startup* tab, and selecting an appropriate *.esp file for an entity type in the *Startup Personal Libraries* section.

***Note:*** By default, when saving an entity of a certain type for the first time to a library, a new *.esp file, with a generic name, such as material.esp, will be automatically created in the user's windows user directory, which is typically similar to the following: "C:\Users\[username]\AppData\Local\Femap\[femap version]". In some organizations, these file may be stored somewhere different, therefore, please consult someone within the organization for confirmation.

### *Shared*

The *Shared* tab actually contains all the same controls as the *Personal* tab, but is somewhat different. First, the libraries which appear on the *Shared* tab for the various entity types must all reside in the same directory. This directory path can be specified via *File, Preference*, choosing the *Library/Startup* tab, then specifying a *Shared Library Path* in the *Startup Shared Libraries* section. Second, it is common for “shared” libraries to reside somewhere, such as a directory on a shared server, where a user’s specific level of permissions may or may not allow the user to use a *Save* button in a creation dialog box for a specific entity type to save to that file

***Note:*** If items cannot be saved from the *Shared* tab, please check with someone in the organization to determine if the user should be allowed to save to the directory in general or the files in the directory.

### *Femap Standard Libraries*

The *FEMAP Standard Libraries* tab offers the ability to load an item from a library file (*.esp file) which has been distributed with FEMAP. These library files reside in the FEMAP installation directory and items cannot be saved to or deleted from these libraries at any time. Also, there is no way to change the path to these libraries.

***Note:*** Values in Library files distributed with FEMAP are believed to be correct, but have not been verified. You must verify these values are correct and appropriate before using them for any purpose.

## *OLE/COM API*

**New and modified API Objects and Attributes**

- Added NasMsnlArcLenOn, NasMsnlArcLenConstr, NasMsnlArcLenMinAlr, NasMsnlArcLenMaxAlr, NasMsnlArcLenMaxR, NasMsnlArcLenScale, NasMsnlArcLenDIter, NasMsnlArcLenMxInc, NasMsnlArcLenInitLdfac, NasMsnlArcLenMxLdfac, NasMsnlArcLenSkipFac, NasMsnlCntTINTMTH, NasMsnlCntBETA, NasMsnlCntGAMA, NasMsnlCntALFA, NasMsnlCntTETA, NasStatsubOn, NasStatsubBuckle, NasStatsubPreload, NasMsnlkCnt2MATSYM, NasMsnlkCnt2INERTIA, NasMsnlkCnt2CRINFAC, NasMsnlkCnt2ITEREF, NasMsnlkCnt2TSDYN, NasMsnlkCnt2RJPZ, NasMsnlkCnt2RJPN, NasMsnlkCnt2TSVSC, NasMsnlkCnt2VSCOTE, and NasMsnlkCnt2VSCOSN attributes to the Analysis Case Object. Also, added NasMsnlkCnt2DTINIT, NasMsnlkCnt2DTMIN, NasMsnlkCnt2DTMAX, NasMsnlkCnt2EQMFMX, NasMsnlkCnt2EQMFMIN, and NasMsnlkCnt2TINTMTH, which were added to support some options which were renamed in Simcenter Nastran.
- Added NasBulkPARAMBuckle, NasMsnlCntTINTMTH, NasMsnlCntBETA, NasMsnlCntGAMA, NasMsnlCntALFA, NasMsnlCntTETA, NasMsnlkCnt2MATSYM, NasMsnlkCnt2INERTIA, NasMsnlkCnt2CRINFAC, NasMsnlkCnt2ITEREF, NasMsnlkCnt2TSDYN, NasMsnlkCnt2RJPZ, NasMsnlkCnt2RJPN, NasMsnlkCnt2TSVSC, NasMsnlkCnt2VSCOTE, and NasMsnlkCnt2VSCOSN attributes to the Analysis Manager Object. Also, added NasMsnlkCnt2DTINIT, NasMsnlkCnt2DTMIN, NasMsnlkCnt2DTMAX, NasMsnlkCnt2EQMFMX, NasMsnlkCnt2EQMFMIN, and NasMsnlkCnt2TINTMTH, which were added to support some options which were renamed in Simcenter Nastran.
- Added UseSolverAxes attribute to the Connection Region Object.
- Added items to the flag and pval attributes of the Connection Property Object (for ABAQUS Thermal options of ABAQUS tab and new options on Multistep Structural tab).
- Added CopyToActiveLayer attribute to the CopyTool Object.
- Added location attribute to the GFXArrow Object.
- Added failuretheory attribute to the Global Ply Object.
- Added failuretheory and vfailuretheory attributes to the Layup Object.
- Added items to the flag and pval attributes of the Property Object (for ABAQUS Thermal options for Gaps).
- Added ShowInModelInfo and SelectVisibleOnly attributes to the Selector Object.

**New and Updated API Methods**

- Updated GetLibrary, PutLibrary, and Delete Library methods on the common entity object to allow selection of the type of library, “Personal”, “Shared”, or “Femap Standard Libraries”.
- Updated GetLibraryOfType method on the Material and Property objects.

- Added SPOPT, PSDUNIT, PFACT, and PSDCOM methods to the Analysis Case Object.
- Added GetUsedFREQS, PutUsedFREQS, SPOPT, PSDUNIT, PFACT, and PSDCOM methods to the Analysis Manager Object.
- Added AddDataColumn and GetColumnInfo2 methods to the Data Table Object.
- Added GetMatlOrientVec as a method to the Element Object
- Added GetFREQType, SetFREQData, GetFREQData, SetFREQ1Data, GetFREQ1Data, SetFREQ2Data, GetFREQ2Data, SetFREQ3Data, GetFREQ3Data, SetFREQ4Data, GetFREQ4Data, SetFREQ5Data, GetFREQ5-Data methods to the Frequency Object. Also, updated Get and Put methods.
- Added AddPly2, InsertPly2, SetPly2, GetPly2, SetAllPly2, and GetAllPly2 methods to the Layup Object. Also, updated HasGlobalPly method.
- Added RemoveNotVisible and AddAllAnalysisCases methods to the Set Object.
- Added SetFontData and GetFontData methods to the Text Object. Also, updated the Get method.
- Added CollectorAppearanceSetFontData method to the User Defined Graphics Object.
- Added SetFontData and GetFontData methods to the View Object. Also, updated ElementNoResultMode to accept an additional value.
- Added Spin method to the View Orient Object.

### Removed API Methods

- Removed AllOn, AllOff, TurnOn, TurnOff, AddFreq, AddFreq1, AddFreq2, AddFreq3, AddFreq4, AddFreq5, AddFreqByFuncID, AddFreq5ByFuncID, UpdFreq, UpdFreq1, UpdFreq2, UpdFreq3, UpdFreq4, UpdFreq5, UpdFreqByFuncID, UpdFreq5ByFuncID from the Frequency Object

### The following functions have been added or updated:

- feGetCurrentLibraryName
- feSetCurrentLibraryName
- feSolidInsideMulti
- feMeshTetSolid3
- feModifyEdit
- feNodesBetweenNodes
- feSolidElementsInARow

### New and updated Global Variables

- Added Pref_ModelThumbnail, Pref_ModelTooltip, PickVisibleOnly, Pref_SharedLibPath, Pref_Geometry-BooleanTolerance, Pref_GraphicsFont, and Pref_GraphicsFontSize to set various preferences.
- Added Pref_SuppressScreenEntities, which can disable selection of all “Active Screen Entities” (i.e. View Legend, View Axis, Post Titles, and Contour Legend) in the graphics window.
- Added PickVisibleOnly option which enables the *Select Visible Only* option on the *Pick^* menu of the standard entity selection dialog boxes and on *Selector Actions* icon menu of the *Select* toolbar.
- Updated Pref_MsgWndFontSize and Pref_MsgWndFontSize to set various preferences.

## *Preferences*

### Database

- Added *Windows Explorer Data* section. This section controls if a thumbnail image and/or general information about the model is stored with a FEMAP model file (*.modfem).

*Save Thumbnail*, enabled by default, will store an image (active view of model when model is saved) with the model, which is displayed with/as the icon for *.modfem file in Windows (File) Explorer.

*Save Size and Notes Info*, enabled by default, stores information including: FEMAP Version number; Number of Solids, Nodes, Elements, and/or Output Sets; and the first 10 lines of the text specified for the model via the *File, Notes* command. This information will be displayed in a tooltip when the cursor is placed over a *.modfem file for a short period of time.

### Geometry/Model

- Added *Solid Boolean Tolerance* option to the *Geometry Preferences* section. Controls the tolerance value the Parasolid geometry kernel uses for Boolean operations. By default, the value is 0.0, which automatically determines a tolerance based on the *Solid Geometry Scale Factor*. **Changing this setting may cause geometry problems. Do not change this setting unless instructed to do so by FEMAP Support**

### Library/Startup

- Moved *Material Type Definition* library to new *Other Libraries* section.
- Updated the name of the *Libraries* section to *Startup Personal Libraries (Filenames Also Set Standard Shared/ System Libraries)*.

This section specifies the library file which will appear, when FEMAP is initially started, in the *Library* drop-down of the *Personal* tab of the *Select From (entity type) Library* dialog box for each entity type which supports the use of libraries. It is possible to use the *Browse* (...) icon button to the right of each field in this section to search for a specific directory where a FEMAP library file might be found. The file name of the library file (*.esp file) will also be used to select the default library selected in the *Library* drop-down on the *Shared* and *Femap Standard Libraries* tabs in the *Select From (entity type) Library* dialog box as well. If a full directory path is not specified, FEMAP will look for the file in "C:\Users\[username]\AppData\Local\Femap\[femap version]".

The *View Library* contains views that can be loaded into the model. This file must exist to be able to use the *Load View* button in the *View Visibility* dialog box or the *Load* button in the *View Manager*. If a library file for views does not exist, clicking *Save View* or *Save* creates a new file here, "C:\Users\[username]\AppData\Local\Femap\[femap version]", when on the *Personal* tab, or in the directory specified for *Shared Library Path* when on the *Shared* tab.

The *Material*, *Property*, *Layup*, *Connection Property*, *Function*, *Analysis*, *Format*, and *Chart* libraries are files which contain data that can be accessed via the *Save* and *Load* buttons on the creation commands (also *List, Output, Formatted*). The name of an existing file must be specified to use the *Load* option for any entity type. If a library file for an entity type does not currently exist, using *Save* will create a new file in this directory, "C:\Users\[username]\AppData\Local\Femap\[femap version]", while on the *Personal* tab. If on the *Shared* tab instead of *Personal*, the new library file is created in the directory specified for *Shared Library Path*.

- Added *Startup Shared Libraries* section.

This section is used to specify the directory path to any libraries which will be available on the *Shared* tab of the *Select From (entity type) Library* for each entity type that supports libraries. If no path is specified, the *Shared* tab will not appear in the various *Select From (entity type) Library* dialog boxes. Any library file to be "shared" should be placed into a single directory. The *Select From (entity type) Library* dialog box for each type of entity will only recognize libraries containing that entity type, so all of library files for the different entity types can reside in the same directory. For example, it is not possible to select a library containing properties using the *Library* drop-down in any tab of the *Select From Material Library* dialog box.

# What's New - 12.0 and 12.0.1

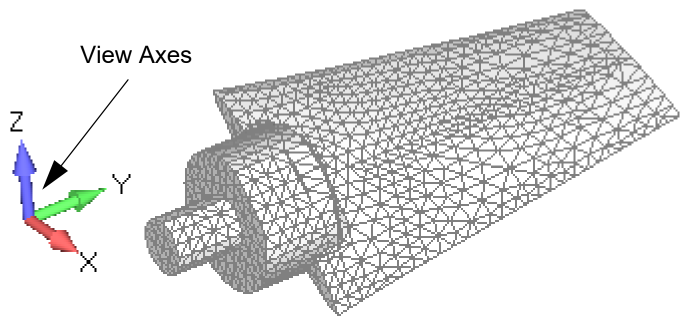
## User Interface

General, Menu, Toolbars, Mesh Point Editor - New for FEMAP 12!, Model Info tree, Meshing Toolbox, PostProcessing Toolbox, Charting pane, Data Surface Editor, Data Table

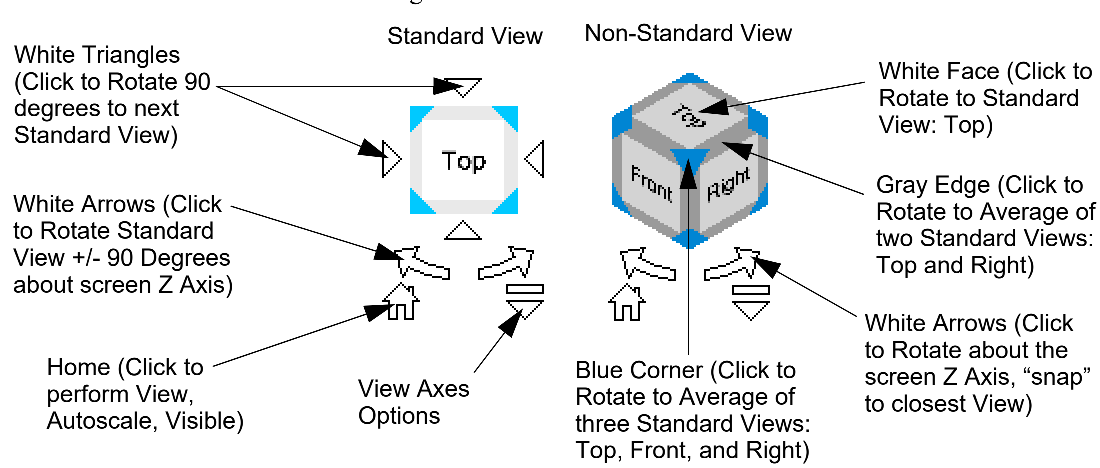
### General

- Added "X" icon on the tab for each view which can be used to quickly close a view or an entire model, but only when a single view is currently visible for the model.
- Added *Previous On* icon button to select Output Sets and/or Output Vectors which where last selected via the *Select Output Set(s)* or *Select Results* dialog boxes found in various commands throughout FEMAP.
- Updated View Axes to display an Orientation Cube when clicked, which can be used to quickly rotate models to "standard views" and other orientations which are the average of two or three different "standard views".

The view axes represent the orientation of the global axes. By default, "1..ID" is selected in the *Show As* section, which will label the axes with X, Y, and Z accordingly, while "0..No Labels" will show the axes with no labels. The *Color/Draw Mode* controls the appearance of the view axis, which can be displayed as lines without arrows, with arrows, or as "Solid" arrows, using either the specified *View Color* or RGB (Red, Green, Blue). The default is "5..RGB Solid"..




The *View Axes* is a "live screen entity". Simply click anywhere in the "view axis area" in the graphics window, even while in a different command, to make the entity "live". Once the entity is live, it can be "dragged" to any location within the current graphics window. In addition, when the entity is live, the view axes changes into a cube, surrounded by various symbol, which can be used to update the of the orientation of the model within the active view to a "standard view" or the average of two or three of the "standard views":



"Standard View" in above figure refers to the model being orientated in one of the "standard orientations" available via the *View, Rotate, Model* command. See Section 6.2.1.1, "View, Rotate, Model..." for more information. Also, the "screen X Axis" starts on the left of the graphics window and extends horizontally to the right, the "screen Y Axis" extends vertically from bottom to top, and the "screen Z Axis" extends from the screen towards the user.

The symbols, along with the faces, edges, and corners of the cube, can be clicked to perform an orientation operation or, in the case of the *View Axes Options* symbol, open a dialog box. Each will be explained in greater detail.

***View Axes Options*** - opens the *View Options* dialog box with *Category* set to *Tools and View Style* and *View Axes* selected in the *Options* list.

***Home*** - simply performs the *View, Autoscale, Visible* command. See Section 6.2.3.3, "View, Autoscale, Visible".

***White Faces of Cube*** - each face of the cube has the name of a standard orientation: *Top*, *Bottom*, *Left*, *Right*, *Front*, and *Back*. Clicking a face will update the orientation of the model to the standard orientation named on the face, in terms of the screen X Axis and screen Y Axis, but may be rotated around the screen Z axis. (Default, +90 degrees, 180 degrees, or -90 degrees). When the text is "right side up" and reads left to right, this is equivalent to using *View, Rotate, Model* and clicking the button with the same name or clicking the corresponding icon on the *View Orient* toolbar. When the text is vertical and reads bottom to top, this is +90 degrees around the screen Z Axis from the standard orientation, vertical and reads top to bottom is -90 degrees, and upside down is 180 degrees.

***Grey Edges of Cube*** - each edge of the cube lies between two of the named faces and there are twelve edges available. Clicking on an edge will update the orientation to the average orientation of the two named faces, with the selected edge appearing vertically or horizontally in the middle of "cube area". If the edge was closer to vertical then horizontal before being clicked, the edge will end up vertical, otherwise it will end up horizontal.

***Blue Corners of Cube*** - each corner of the cube is the location were a corner from three different named faces converge and there are 8 corners available. Clicking on a corner will update the orientation to the average orientation of the three named faces, with the selected corner appearing in the middle of "cube area". When the *Top* face is to the lower left of the blue corner, the *Right* face to the lower right, and the *Back* face on top, this corresponds to clicking the *Isometric* button in *View, Rotate, Model* or clicking the *Isometric* icon on the *View Orient* toolbar.

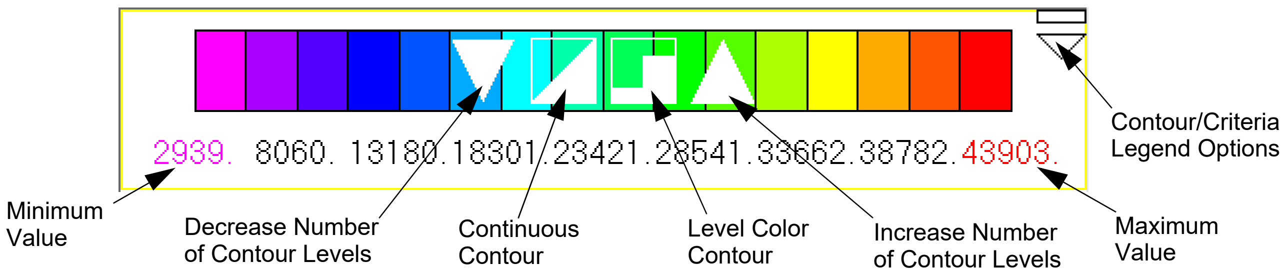
***White Triangles*** - only visible when the model is in, or very close to, a standard orientation and appear to the left, right, top, and bottom of the cube. When the triangle to the left of the cube is clicked, the model is rotated +90 degrees around the screen Y Axis, while clicking the one on the right will rotate the model -90 degrees. When the triangle above the cube is clicked, the model is rotated +90 degrees around the screen X Axis, while clicking the one below the cube will rotate the model -90 degrees. In addition, the model will always end up in a standard orientation, even if it was not before the

***White Arrows*** - always appear below the cube, one to the lower left, pointing left and the other to the lower right, point right. If the current model orientation corresponds with an orientation which is attained by clicking on a face or edge of the cube, then clicking the arrow to the lower right will simply rotate the model +90 degrees around the screen Z Axis, while clicking the arrow to the lower left will rotate the model -90 degrees. If the current orientation was attained by clicking a corner of the cube, then clicking the arrow to the lower right will rotate the model +60 degrees around the screen Z axis, while clicking the other arrow will rotate the model -60 degrees. If the model is not in any of the orientations mentioned above, then it will be rotated to the nearest orientation which can be attained by clicking on a face, edge, or corner of then cube, then rotated either +/-90 degrees or +/- 60 degrees around the screen Z Axis, following the rules mentioned above.

Typically, the view axes are displayed in the lower left corner of the view, but a new position can be specified by pressing *Position*, which displays the *Select View Position - View Axes* dialog box. The *X* and *Y* positions are the relative distances, as a percentage of overall screen size, from the view axes to the upper left hand corner of the view where the view axes are being displayed. For instance, a value of *X* = 0 and *Y* = 0 positions the view axes as far to the left and as close to the top of the view as possible, while X = 100 and Y = 100 positions the view axes as for to the right and as close to the bottom as possible. To select a new position graphically, simply click a position in the graphics window and click OK.

- Updated Contour Legend to be an interactive screen entity.

The *Contour/Criteria Legend* is a "live screen entity". When not engaged in a command, simply click anywhere in the "legend area" in the graphics window to make the entity "live". Once the entity is live, it can be "dragged" to any location within the current graphics window. In addition, a number of symbols and items which can be clicked to modify the legend will appear on or near the legend:

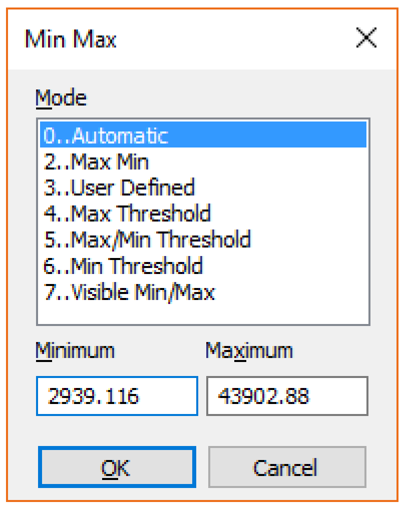



Clicking the *Increase Number of Contour Levels* and the *Decrease Number of Contour Levels* symbol adds or removes a contour level from the legend

***Note:*** Clicking the symbols to increase or decrease the number of contour levels only works when the *Contour Palette* option in *Contour/Criteria Levels* is set to "0..Standard Palette", as the number of colors in the User Palette always set the number of levels when *Contour Palette* is set to "1..User Palette". See Section 8.3.10, "Contour/Criteria Levels..." for more information.

Clicking the *Continuous Contour* symbol sets the *Contour Fill Mode* option for *Contour Type* to "0..Continuous", while clicking the *Level Color Contour* symbols set the *Contour Fill Mode* option to "1..Level Colors".

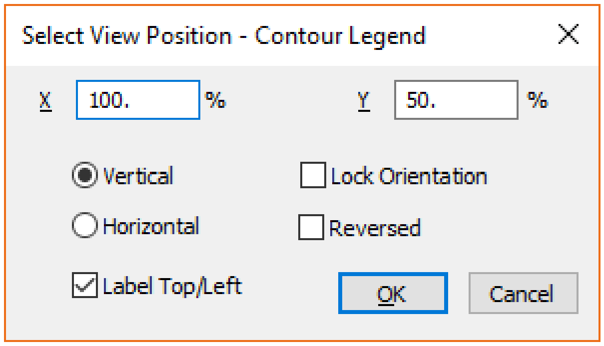
Clicking either the *Minimum Value* item or *Maximum Value* item will open the *Min Max* dialog box:




This dialog box can be used to specify the *Level Mode* for *Contour/Criteria Levels* and potentially specify *Minimum* and/or *Maximum* values to be used by the selected *Mode*. For more information about the various *Level Mode* options, see Section 8.3.10, "Contour/Criteria Levels...". If the *Minimum Value* item is clicked, the *Minimum* field will be active and a value can be entered without clicking into the field, while clicking the *Maximum Value* item does the same thing for the *Maximum* field.

Clicking the *Contour/Criteria Legend Options* symbol (i.e., "down arrow" symbol) will open the *View Options* dialog box with *Category* set to *PostProcessing* and *Contour/Criteria Legend* selected in the *Options* list.

There are 18 options in the *Position* list: Options 0-7 display the legend in a "standard" position with the values in "default" order (i.e., lowest to highest from bottom to top or left to right); while Options 9-16 display the legend in a "standard" position with values in "reversed" order (i.e., lowest to highest from top to bottom or right to left). The other two options, "8..Custom" and "17..Custom, Reversed", show the legend in a "custom" position with the values in "default" order or "reversed" order, respectively. To specify a "custom" position via the *View Options* dialog box, click the *Custom Position...* button to display the *Select View Position - Contour Legend* dialog box:

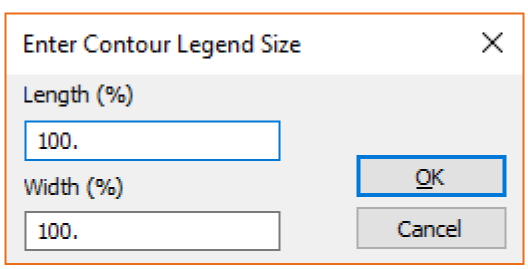



The *X* and *Y* positions are the relative distances, as a percentage of overall screen size, from the upper left hand corner of the "legend area" (area encompassing the legend and values) to the upper left hand corner of the view where the legend is being displayed. For instance, a value of *X* = 0 and *Y* = 0 positions the legend as far to the left and as close to the top of the view as possible, while X = 100 and Y = 100 positions the legend as for to the right and as close to the bottom as possible. In addition, the legend can have either a *Vertical* or *Horizontal* orientation, which can be "locked" by enabling the *Lock Orientation* option. Finally, to display the values in "reversed" order, enable the *Reversed* option and/or enable *Label Top/Left* to have the labels always appear to the left of a vertical legend/ above a horizontal legend (when option is disabled, appear right of a vertical legend/below a horizontal legend).

All of he "standard" options for *Position* simply specify specific values and options in the *Select View Position - Contour Legend* dialog box, which are detailed in the following table (ID of corresponding option which has the *Reversed* option enabled in parentheses):

| Option (ID of Reversed Option) | X % | Y % | Orientation | Label Top/Left |
|---|---|---|---|---|
| 0..Top Left (9) | 0.0 | 0.0 | Vertical | Disabled |
| 1..Top Center (10) | 50.0 | 0.0 | Horizontal | Disabled |
| 2..Top Right (11) | 100.0 | 0.0 | Vertical | Enabled |
| 3..Center Left (12) | 0.0 | 50.0 | Vertical | Disabled |
| 4..Center Right (13) | 100.0 | 50.0 | Vertical | Enabled |
| 5..Bottom Left (14) | 0.0 | 100.0 | Vertical | Disabled |
| 6..Bottom Center (15) | 50.0 | 100.0 | Horizontal | Enabled |
| 7..Bottom Right (16) | 100.0 | 100.0 | Vertical | Enabled |

The *Legend Size...* button displays the Enter Contour Legend Size dialog box:



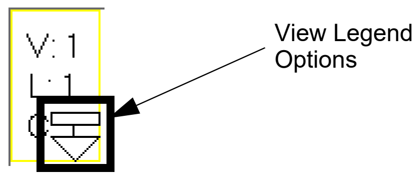

The *Length (%)* option controls the distance from top to bottom when the legend has a vertical orientation or left to right when the legend has a horizontal orientation. This option can only be used to "shrink" the overall length of the legend and must have a value between 1 and 100 (default). For example, specifying a value of 50 will shrink the legend to 50% of the default length.

The *Width (%)* option controls the thickness of only the color bar portion of the legend, which is left to right when the legend has a vertical orientation or top to bottom when the legend has a horizontal orientation. This option can be used to make the overall width of the color bar thicker or thinner and must have a value between 10 and 200 (default is 100). For example, specifying a value of 50 will display the color bar at half the normal width, while specifying a value of 150 will display the color bad at 1.5 times the normal width.

If desired, the *Length (%)* and *Width (%)* options can both be specified at the same time to create a "shrunken" legend which is either thicker or thinner, then positioned accordingly using any of the other available options.

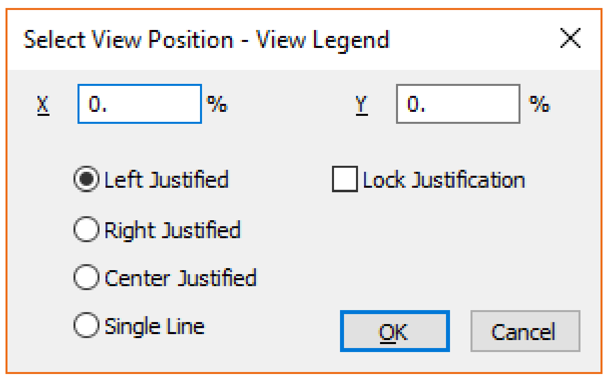
- Updated View Legend to be an interactive screen entity.

The *View Legend* is a "live screen entity". When not engaged in a command, simply click anywhere in the "view legend area" in the graphics window to make the entity "live". Once the entity is live, it can be "dragged" to any location within the current graphics window. In addition, when the entity is live, the *View Legend Options* symbol (i.e., "down arrow" symbol) appears:




...which when clicked, will open the *View Options* dialog box with *Category* set to *Tools and View Style* and *View Legend* selected in the *Options* list.

There are 9 options in the *Position* list: Options 0-7 display the view legend in a "standard" position, while "8..Custom" can be used to place the view legend in a "custom" position. To specify a "custom" position via the *View Options* dialog box, click the *Custom Position...* button to show the *Select View Position - View Legend* dialog box:




The *X* and *Y* positions are the relative distances, as a percentage of overall screen size, from the upper left hand corner of the "view legend area" to the upper left hand corner of the view where the view legend is being displayed. For instance, a value of $X = 0$ and $Y = 0$ positions the view legend as far to the left and as close to the top of the view as possible, while X = 100 and Y = 100 positions the view legend as for to the right and as close to the bottom as possible. In addition, the text within the "view legend area" can be *Left Justified*, *Right Justified*, or *Center Justified*. There is also an option to show all text on a *Single Line*, but be careful when using this option, as the graphics window needs to be wide enough to display the entirety of the text, otherwise the beginning and end may not be visible. Finally, The justification can also be "locked" by enabling the *Lock Justification* option.

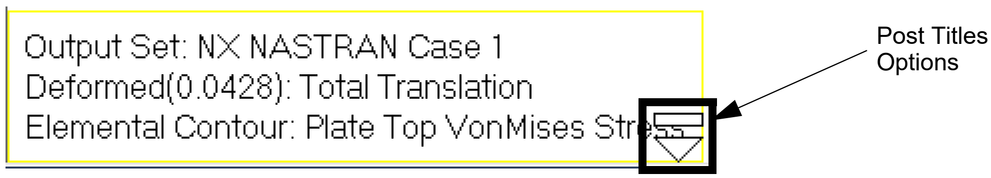
All of the "standard" options for *Position* simply specify specific values and options in the *Select View Position - View Legend* dialog box, which are detailed in the following table:

| Option | X % | Y % | Justification |
|---|---|---|---|
| 0..Top Left | 0.0 | 0.0 | Left Justification |
| 1..Top Center | 50.0 | 0.0 | Center Justification |
| 2..Top Right | 100.0 | 0.0 | Right Justification |
| 3..Center Left | 0.0 | 50.0 | Left Justification |
| 4..Center Right | 100.0 | 50.0 | Right Justification |
| 5..Bottom Left | 0.0 | 100.0 | Left Justification |
| 6..Bottom Center | 50.0 | 100.0 | Center Justification |
| 7..Bottom Right | 100.0 | 100.0 | Right Justification |

**Note:** If opening a model file saved in version 11.4.2 or earlier and *Position* was set to “1..Top Center” or “6..Bottom Center”, then *Single Line* will be selected and the *Lock Justification* option will be enabled. If desired, simply click the *Custom Position...* button to change these options. This will also happen if importing a FEMAP neutral file exported in version 11.4 format or earlier.

- Updated Post Titles to be an interactive screen entity.

The *Post Titles* are a “live screen entity”. When not engaged in a command, simply click anywhere in the “post titles area” in the graphics window to make the entity “live”. Once the entity is live, it can be “dragged” to any location within the current graphics window. In addition, when the entity is live, the *Post Titles Options* symbol (i.e., “down arrow” symbol) appears:

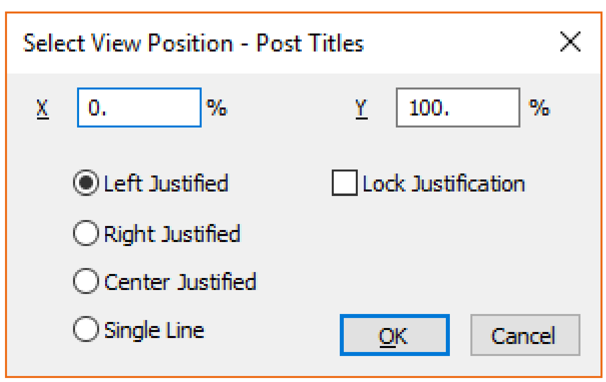



...which when clicked, will open the *View Options* dialog box with *Category* set to *PostProcessing* and *Post Titles* selected in the *Options* list.

By default, the justification will automatically change from *Left Justification* to *Center Justification* to *Right Justification* as the *Post Titles* entity is dragged from the left side of the graphics window to right. To have the justification remain the same as the entity is dragged around the graphics window, simply click the “down arrow” symbol, click the *Custom Position...* button, and enable the *Lock Justification* option, then click OK to all dialog boxes.

This legend contains information about the output set and output vectors being displayed, with additional options to include the current averaging options, min/max values, and/or the full directory path and file name, along with time and date information, for the output file used to create displayed results. For *Legend Style*, choose from “0..IDs Only”, “1..Titles” (Default), “2..Titles and Average Data”, “3..Titles and Min/Max Data”, “4..Titles and Min/Max/Average”, “5..Titles, File Name and Date”, “6..Titles/File/Average”, 7..Titles/File/Min/Max”, or “8..Titles/File/Min/Max/Average”. In order to see “Max/Min” information, *Label Mode* for the *Contour/Criteria Style* option must be set to “2..Max Min”, “3..Max Only”, or “4..Min Only” (see Section 8.3.9, "Contour/Criteria Style...").

There are 9 options in the *Position* list: Options 0-7 display the post titles in a “standard” position, while “8..Custom” can be used to place the post titles in a “custom” position, To specify a “custom” position via the *View Options* dialog box, click the *Custom Position...* button to display the *Select View Position - Post Titles* dialog box:




The *X* and *Y* positions are the relative distances, as a percentage of overall screen size, from the upper left hand corner of the “post titles area” to the upper left hand corner of the view where the post titles are being displayed. For instance, a value of $X = 0$ and $Y = 0$ positions the post titles as far to the left and as close to the top of the view as possible, while X = 100 and Y = 100 positions the post titles as for to the right and as close to the bottom as possible. In addition, the text within the “post titles area” can be *Left Justified*, *Right Justified*, or *Center Justified*. There is also an option to show all text on a *Single Line*, but be careful when using this option, as the graphics window needs to be wide enough to display the entirety of the text, otherwise the beginning and end may not be visible. Finally, The justification can also be “locked” by enabling the *Lock Justification* option.

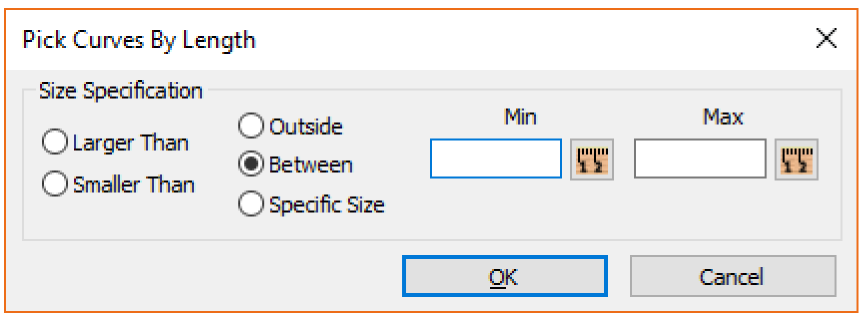
All of the “standard” options for *Position* simply specify specific values and options in the *Select View Position - Post Titles* dialog box, which are detailed in the following table:

| Option | X % | Y % | Justification |
|---|---|---|---|
| 0..Top Left | 0.0 | 0.0 | Left Justification |
| 1..Top Center | 50.0 | 0.0 | Center Justification |
| 2..Top Right | 100.0 | 0.0 | Right Justification |
| 3..Center Left | 0.0 | 50.0 | Left Justification |
| 4..Center Right | 100.0 | 50.0 | Right Justification |
| 5..Bottom Left | 0.0 | 100.0 | Left Justification |
| 6..Bottom Center | 50.0 | 100.0 | Center Justification |
| 7..Bottom Right | 100.0 | 100.0 | Right Justification |

***Note:*** If opening a model file saved in version 11.4.2 or earlier and *Position* was set to “1..Top Center” or “6..Bottom Center”, then *Single Line* will be selected and the *Lock Justification* option will be enabled. If desired, simply click the *Custom Position...* button to change these options. This will also happen if importing a FEMAP neutral file exported in version 11.4 format or earlier.

- Added “By Size” option in the “Pick^” menu of the standard entity selection dialog box to select Curves by Length, Surfaces by Area, and Solids by Volume using the desired Size Specification option.

Only available when selecting curves, surfaces, or solids. Curves are selecting by Length, Surfaces by Area, and Solids by Volume.




The same options exist for each type of geometric entity: *Larger Than* a maximum value, *Smaller Than* a minimum value, *Outside* or *Between* two values, or a *Specific Size* within a specified *Tolerance*.

- Added “Add Connected Tangent Curves” option in the “Pick^” menu of standard entity selection dialog box and is only available when selecting curves. Using the *Add Connected Tangent Curves* command allows you to quickly add “connected tangent curves” to the selection list by first selecting any number of curves. This is a helpful picking tool when using the *Geometry, Curve - From Surface, Offset Curves/Washer* command.
- Added ability to use “ALT+S” and “ALT+E” in Standard Entity Selection dialog boxes to choose “Select All” and “Reset”, respectively.
- Updated “Property Data” to be “Element / Property Data” in the Contour Model Data dialog to reflect the fact that some of the quantities are specified as Element Data.

## Menu

- Added *Tools, Mesh Point Editor* and *Tools, Report Generator* commands to the *Tools* menu. See Mesh Point Editor - New for FEMAP 12! and the Tools section for more information. In addition, removed the *Tools, Stress Wizard* command from the *Tools* menu. If desired, it can be added to the User Interface by using the *Tools, Toolbars, Customize* command, clicking the *Commands* tab, choosing *Additional Commands* from the list of *Categories*, then dragging the *Stress Wizard* command onto any toolbar or menu.
- Added *Geometry, Curve - Centerline* command to the *Geometry* menu. In addition, updated *Geometry, Surface, Ruled* to *Geometry, Surface, Ruled/Between Curves* to reflect new functionality in the command. Finally, removed the *Geometry, Volume* submenu. See Geometry section for more information.
- Added *Connect, Connector Set* command to the *Connect* menu. See Connections (Regions, Properties, and Connectors) section for more information.

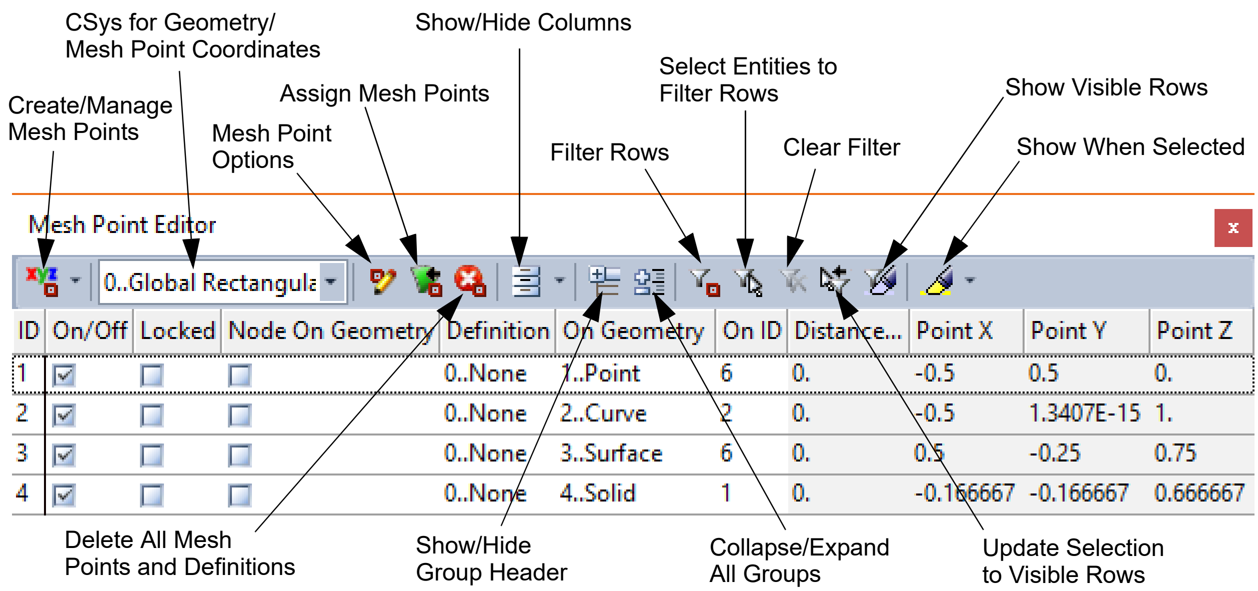
- Added *Model, Optimization* submenu, which contains four commands: *Model, Optimization, Variables and Topology Regions*; *Model, Optimization, Limits*; *Model, Optimization, Manufacturing Constraints*; and *Model, Optimization, Update from Punch*. See Optimization section for more information.
- Added *Mesh, Editing, Cohesive Meshing* command to the *Mesh* menu. In addition, added *Mesh, Edge/Skin Elements* submenu, which contains two commands: *Mesh, Edge/Skin Elements, Line Elements on Edges* and *Mesh, Edge/Skin Elements, Planar Elements on Faces*. Previously, these two commands were a single command, *Mesh, Edge Members*, but were split into two commands for clarity. Finally, updated the *Mesh, Mesh Control, Mesh Points on Surface* command to toggle the *Mesh Point Editor* dockable pane on/off, instead of launching a command. See Meshing and Mesh Point Editor - New for FEMAP 12! sections for more information.
- Added *Modify, Renumber, Mesh Point* and *Modify, Update Other, Align Surface Parameters* commands to the *Modify* menu. In addition, added *Modify, Reflect* submenu, which contains seven commands: *Modify, Reflect, Point*; *Modify, Reflect, Curve*; *Modify, Reflect, Surface*; *Modify, Reflect, Solid*; *Modify, Reflect, Coord Sys*; *Modify, Reflect, Node*; and *Modify, Reflect, Element*. Finally, the *Modify, Project* submenu was moved to its own menu group and the words "To Location" was added to the three commands on the *Modify, Move To* submenu. See Geometry and Meshing sections for more information
- Added *List, Geometry, Mesh Point* command to the *List* menu. See Listing section for more information.
- Added *Delete, Geometry, Mesh Point*; *Delete, Model, Optimization Variable/Topology Region*; *Delete, Model, Optimization Limit*; and *Delete, Model, Manufacturing Constraint* commands to the *Delete* menu.
- Added *Group, Operations, Generate From Criteria* command to *Group* menu. In addition, added *Group, Mesh Point* submenu, which contains seven commands: *Group, Mesh Point, ID*; *Group, Mesh Point, Color*; *Group, Mesh Point, Layer*; *Group, Mesh Point, on Point*; *Group, Mesh Point, on Curve*; *Group, Mesh Point, on Surface*; and *Group, Mesh Point, in Solid*. See Groups and Layers section for more information.
- Added *View, Advanced Post, Dynamic Criteria* and *View, Autoscale, Fit* commands to the *View* menu. See the Views section for more information.
- Added *Help, Dockable Panes, Mesh Point Editor* and *Help, Femap Product Excellence* to the *Help* menu.

### *Toolbars*

- Added *By Size* command to *Selector Actions* icon menu on *Select* toolbar. Only available when Curve, Surface, or Solid is the active entity type in the *Selector Entity* icon menu. Curves can be selecting by Length, Surfaces by Area, or Solids by Volume.
- Added *Connector Set* command to context-sensitive menu when Connector is the active entity type in the *Selector Entity* icon menu of the *Select* toolbar, which allows creation of a Connector Set containing any number of Connectors.
- Updated *Mesh Size* item on the *View Style* icon menu on the *View* and *View - Simple* toolbars to be *Mesh Size/Locations*, as it now toggles visibility of both mesh size on curves and mesh points.

## *Mesh Point Editor - New for FEMAP 12!*

The *Mesh Point Editor* dockable pane provides you with a tool to interactively create, edit, and/or manage mesh points ("hard points") using an intuitive table control. Much like the *Data Table*, each Mesh Point appears as a single row separated into a number of columns when it enters the *Mesh Point Editor.* Also, once in the *Mesh Point Editor*, information about the mesh Points can be sorted, filtered, and evaluated to help you understand each Mesh Point to a greater degree. Unlike the *Data Table*, the *Mesh Point Editor* allows you to modify certain aspects of each Mesh Point in the model, including individual field modification in certain columns.




*Geometry X, Geometry Y, and Geometry Z columns hidden for this figure

All Mesh Points in the model will automatically appear in the *Mesh Point Editor* pane. As new Mesh Points are added to the model, they will automatically be added as a row into the *mesh Point Editor*. If there are a large number of Mesh Points in the model, use the filtering functionality to reduce the table to a more manageable size.

When the *Mesh Editor* pane is floating, it can be repositioned and resized, just as any other floating dockable pane. The *Mesh Point Editor* can be closed by clicking the “X” on the right side of the Title bar. To reopen the *Mesh Point Editor*, use the *Tools, Mesh Point Editor* or the *Mesh, Mesh Control, Mesh Points on Surface* command.

### *Mesh Point Overview*

Mesh Point, which are also sometimes called mesh “hard points” define specific locations on or in geometric entities where nodes will be created when the geometry is meshed. Every Mesh Point is associated to an actual geometric point, which may or may not be at the same location as the Mesh Point. In all cases, the geometric entity is meshed without considering the hard points, then the closest node to each hard point is moved to the hard point location, then the mesh is “re-smoothed”, which typically creates a higher quality mesh.

***Hint:*** This technique works very well when there are a significant number of nodes in the mesh (and a reasonably fine mesh) in comparison to the number of hard points. If there are not many more nodes than hard points, this technique will not produce good meshes. For this type of mesh, it is best to imprint a curve or surface on the surface (see Section 3.2.5, "Curves from Surfaces"), then define the mesh size on that curve or surface. The surface mesher will use a different technique, resulting in a better mesh.

The *Mesh Point Editor* can be used to create Mesh Points either individually, or in bulk, using a number of different methods which will be explained in detail later in this section.

### Mesh Point Editor Rows and Columns

All data from each Mesh Point is sent to the *Mesh Point Editor* in a single row which is partitioned into columns for accurate viewing, sorting, and filtering purposes. When a column header is clicked, an arrow pointing either up or down will appear in the header. When the arrow is pointing up, the data in that column will be sorted from lowest to highest numerical value or alphabetical, from A-Z then a-z again (All UPPER CASE letters come before any lower case letters) depending on the type of data in the column. When the arrow is pointing down, the column will be sorted from highest to lowest numerical value or alphabetical, from z-a then Z-A again. Clicking on a column header will change the direction of the arrow and the reverse the listing.

Column headers can be “dragged and dropped” into new positions to modify the look of the table. Any column header can be “dragged” until a “X” appears over the header, making that column no longer visible. Column width can also be adjusted by placing the cursor on the line between the column headers (this will bring up a two headed arrow), then clicking and dragging the column to the desired width.

Columns can be turned on/off one at a time using the *Show/Hide Columns* icon menu. Also, multiple changes to the columns can be made at once using the *Update Multiple Columns* command on the *Show/Hide Columns* icon menu.

Here is a summary of the available columns:

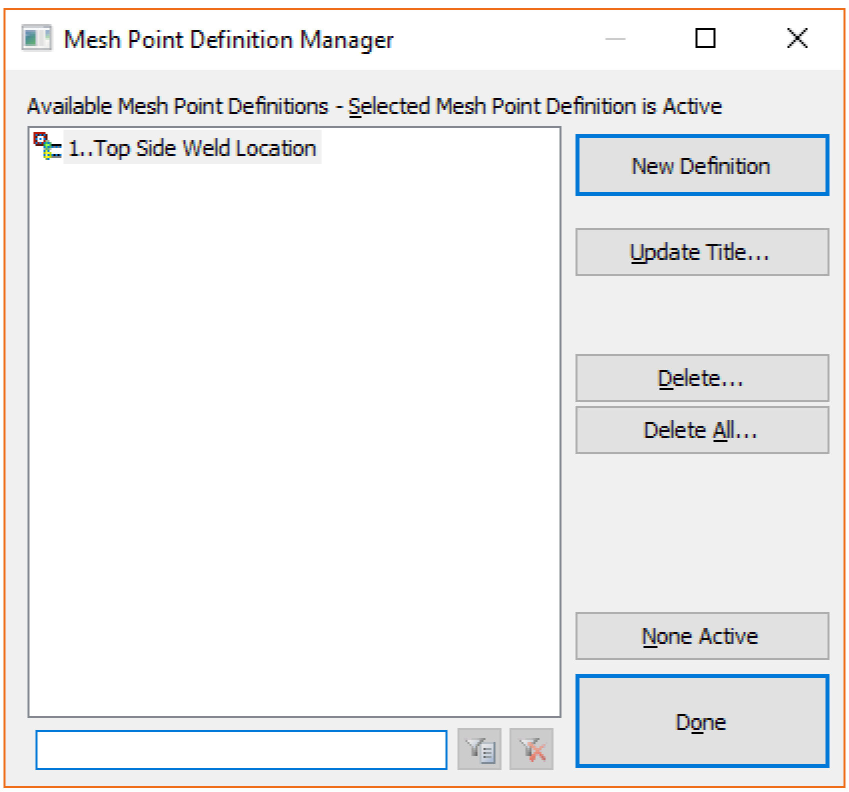
**ID** - ID of the Mesh Point. Cannot be modified directly in the table. This is the ID of the Mesh Point entity itself, not the underlying geometric point which is referenced by each mesh point.

**On/Off** - Interactive toggle displays current status of Mesh point, “On” (checked) or “Off” (not checked) which can be modified in the table. When “On”, the Mesh Point is considered during meshing, while it is not considered during meshing when “Off”. When a Mesh Point is “Off”, it will appear “grayed out” in the *Mesh Point Editor.*

**Locked** - Interactive toggle displays if Mesh Point is “Locked” (checked) or “Unlocked” (not checked), which can be modified in the table. When “Locked”, the Mesh Point will remain in its current location, even if the underlying geometric point it is associated with is moved, while it will move with the underlying point when “Unlocked”.

**Nodes On Geometry** - Interactive toggle displays if the node assigned to a Mesh Point will be projected onto the geometric entity listed in the *On Geometry* column (checked) or not projected onto geometry (not checked), which can be modified in the table. If node is not projected onto geometric entity, it will remain at the location of the Mesh Point’s underlying geometric point.

**Definition** - Current “Mesh Point Definition” associated with Mesh Point. By default, this will be “0..None”, which means the Mesh Point is not associated with a mesh point definition. At the time, the only items which can be defined in a mesh point definition are an *ID* and a *Title*. A mesh point definition can only be created by clicking the “Create/Manage Mesh Point Definition” icon button to the right of the drop-down list, which displays the *Mesh Point Definition Manager*:




The *Mesh Point Definition Manager* can be used to create a new mesh point definition, update the title of existing mesh point definition, delete the active mesh point definition, delete all mesh point definitions, or have no mesh point definition be currently active.

To reduce the number of mesh point definitions being displayed in the *Available Mesh Point Definitions...* list, enter text into the field below the list, then click the *Title Filter* button and only those mesh point definitions that contain the specified text will remain in the list. If additional text is entered and the *Filter* icon button is clicked again, the number of mesh point definitions in the list may be further reduced. To display all available mesh point definitions in the list again, click the *Clear Title Filters* icon button.

Once a mesh point definition has been created and is active, it will be assigned to every new Mesh Point which is created. To change the mesh point definition for a Mesh Point, select a different one from the drop-down list. To remove a mesh point definition from a Mesh Point, choose “0..None” from the drop-down list.

**Color** (column not displayed by default) - Current Color ID, in terms of the FEMAP Color Palette, of Mesh Point. The *Color* field can be modified by inputting a new Color ID and pressing Enter/selecting another row or by clicking the “Color Palette” icon button to the right of the field, then selecting a color from the *Color Palette* dialog box.

**Layer** (column not displayed by default) - Current Layer of Mesh Point. Field contains the Layer ID and can be modified by selecting a layer from the drop-down list or by clicking the “Select Layer” icon button to the right of the drop-down, then selecting a layer from the *Select Layer* dialog box.

**On Geometry** - When set to “0..None”, the Mesh Point is not associated with any geometric entity and the text in the row will be shown in the color red. When set to one of the other options, “1..Point”, “2..Curve”, “3..Surface”, or “4..Solid”, the Mesh Point is associated with a geometric entity of that type, with the ID of geometric entity found in the *On ID* column.

**On ID** - ID of geometric entity to which the Mesh Point is associated. The value in this column corresponds to the geometric entity type specified in the On Geometry column (i.e., when *On ID* is 6 and *On Geometry* is set to “2..Curve”, this refers to Curve 6).

**Distance To Geometry** - Distance from the underlying geometric point associated with Mesh Point to the closest point on the geometric entity specified using the combination of the *On Geometry* and *On ID* columns.

**Geometry X, Geometry Y, Geometry Z** - The XYZ Coordinates of the Mesh Point.

**Point ID** (column not displayed by default) - ID of the underlying geometric point associated to the Mesh Point.

**Point X, Point Y, Point Z** - The XYZ Coordinates of the underlying geometric point associated to the Mesh Point.

### Mesh Point Editor Icons

**Create/Manage Mesh Points Menu** - Contains commands used to create Mesh Points, individually or in bulk.

***At Coordinates*** - creates a new Mesh Point, along with an underlying geometric point, at coordinates specified via the standard coordinate location dialog box.

***On Points*** - creates a new Mesh Point on each geometric points selected using the standard entity selection dialog box, using each selected point as the underlying geometric point for the Mesh Point at the same location.

***On Nodes*** - creates a new Mesh Point, along with an underlying geometric point, at the location of each node selected using the standard entity selection dialog box.

***Load From File*** - displays the *Read Mesh Points File* dialog box, which can be used to select a file containing XYZ Coordinates to use to create new Mesh Points. In some cases, the file may contain additional information to aid in creation of Mesh Points.

***Save To File*** - displays the Save Mesh Points As dialog box, which can be used to save a file containing data, including XYZ Coordinates, for all of the Mesh Points currently in the model. The format of the data saved to the file depends on the type of filed being saved. Choose from “Tab Delimited Text (*.txt)”, “Comma-Separated Text (*.csv)”, “Rich Text Format (*.rtf)”, or the FEMAP-specific “Mesh Point File (*.txt)”.

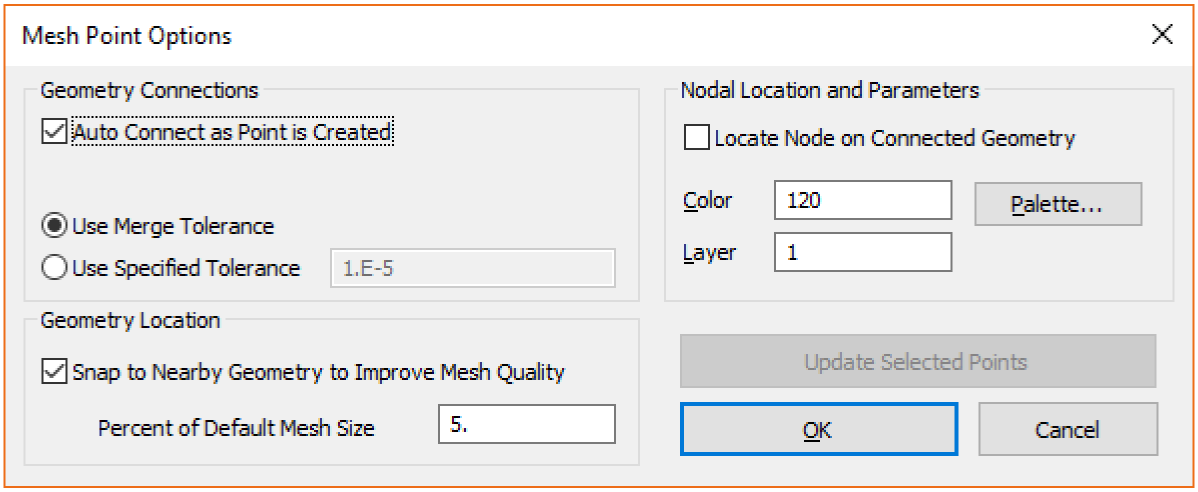
***Copy to Clipboard*** - copies all Mesh Point data to the clipboard, in a specific format, which can then be pasted back into the same FEMAP model or a different model as long as the data remains on the clipboard in the format created by the command.

***Paste from Clipboard*** - when Mesh Point data created by the *Copy to Clipboard* command exists on the clipboard, this command will be available to paste this data into the *Mesh Point Editor*.

***Note:*** Once copied to the clipboard, the data can also be pasted into another program like Microsoft Excel. If the data which was pasted into Excel is then copied back onto the clipboard, it is no longer in the format FEMAP expects, thus, the *Paste from Clipboard* command will not be available.

**CSys for Geometry/Point Coordinates** - This drop-down allows selection of the coordinate system used to display the coordinates in the *Geometry X*, *Geometry Y*, *Geometry Z*, *Point X*, *Point Y*, and *Point Z* columns.

**Mesh Point Options** - Clicking this icon displays the *Mesh Point Options* dialog box:




Controls which options will be on by default for all newly created Mesh Points, until the options are changed via this dialog box, then OK is selected.

***Note:*** This dialog box can also be used to update multiple options for the Mesh Points currently highlighted in the *Mesh Point Editor*, by clicking the *Update Selected Points* button instead of OK.

***Geometry Connections*** - When enabled, *Auto Connect as Point is Created* will automatically attach newly created Mesh Points to geometry, but automatic attachment will only occur if the Mesh Point is within a specified distance, which is determined automatically by the Merge Tolerance (*Use Merge Tolerance*) or specified by the user (*Use Specified Tolerance*). When disabled, newly created Mesh Points will not be attached to any geometry.

***Geometry Location*** - When enabled, *Snap to Nearby Geometry to Improve Mesh Quality* will automatically move newly created Mesh Points to the nearest surface or curve, when the specified Mesh Point is within a distance determined by the value specified for *Percent of Default Mesh Size*.

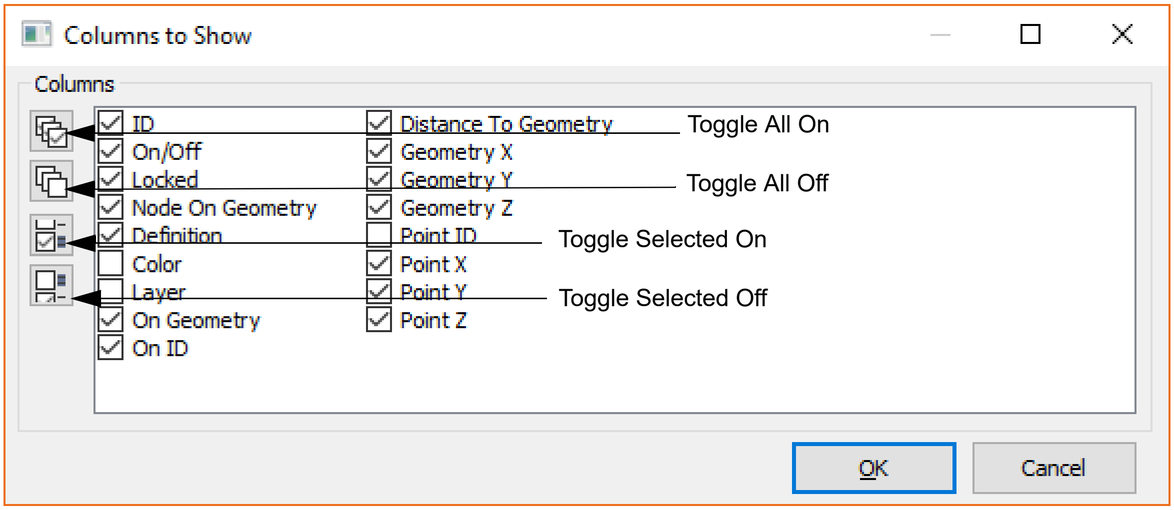
***Nodal Location and Parameters*** - When enabled, *Locate Node on Connected Geometry* will automatically turn on the *Node on Geometry* option for all newly created Mesh Points. To specify a new default color for Mesh Points, either enter the ID of a color into the *Color* field of click the *Palette* button to select a color from the FEMAP Color Palette. To specify a new default layer for Mesh Points, simply enter an ID in the *Layer* field.

**Assign Mesh Points** - When a Mesh Point is within the tolerance specified in the *Geometry Connections* section of the *Mesh Point Options* dialog box, automatically associates that Mesh Point to the closest piece of geometry.

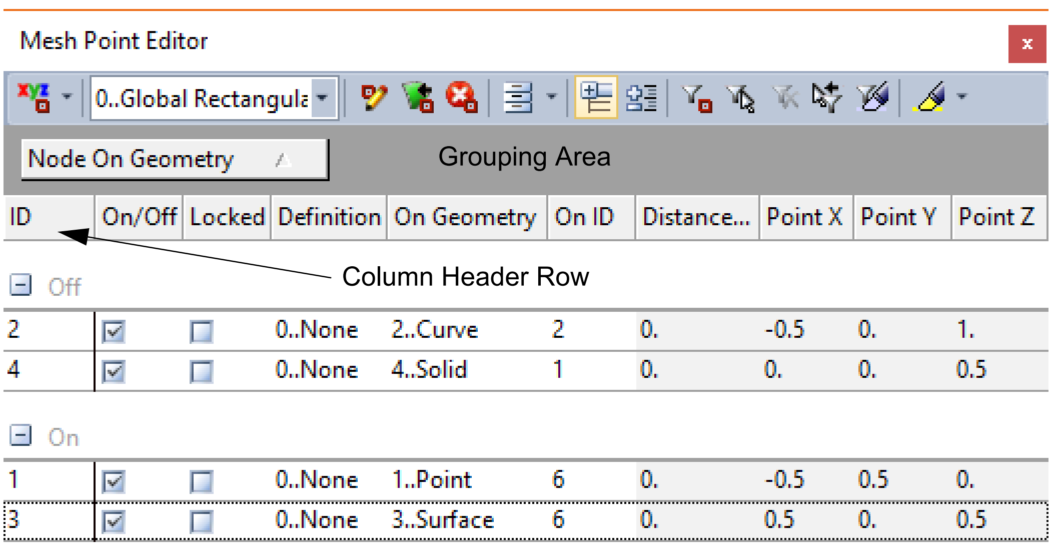
**Delete All Mesh Points and Definitions** - Used to delete all Mesh Points and Mesh Point Definitions in the model.

H**ide/Show Columns Menu** - Clicking the down arrow in this icon displays a menu with commands and a listing of all available columns (visible columns are marked with a check mark in a box colored differently than the menu). By highlighting a column and clicking the mouse, the column can be toggled from hidden to visible and vice versa. This allows you to hide or show one column at a time.

***Update Multiple Columns*** - Brings up the *Columns to Show* dialog box which allows you to check and uncheck multiple column headers at one time. You can also highlight any number of headings in the selection area. Holding down the “Ctrl” key will enable highlighting of multiple output sets. Holding down “Shift” while picking a “first”, then a “last” output set will highlight a range of output sets. Once highlighting is completed, then click *Toggle Selected On* or *Toggle Selected Off* icon buttons to check or uncheck columns. After you are done selecting the columns you want, click *OK* to view your modified *Mesh Point Editor*




**Show/Hide Group Header** - When on (icon colored differently than the toolbar) the *Mesh Point Editor* allows you to drag column headers up into a specified area. Once there, the *Mesh Point Editor* will partition the rows of Mesh Point data using a specific group header or set of group headers. This allows you to "group" Mesh Points in the table with similar characteristics (such as the same option for *Node On Geometry*, etc.) together for a specific purpose. The grouping is multi-level, therefore the Mesh Points are broken down by the first group header, then the second, and so on until the final group header. You can always drag a column header from the "Grouping Area" back to the "Column Header Row" and the remaining group levels will keep their positions in the hierarchy.

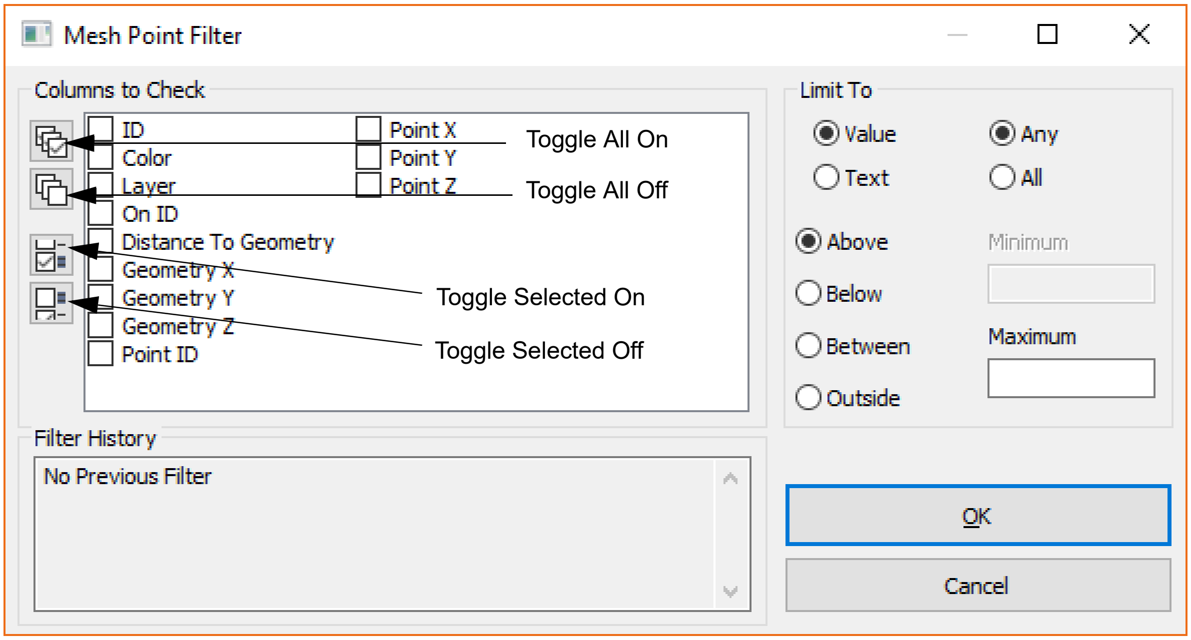



***Note:*** The expand/collapse toggles allow you to collapse and expand individual categories and sub-categories in the groups. When the toggle shows a "+" it means the group category can be expanded, when it shows a "-" it means the group category can be collapsed.

**Expand/Collapse All Groups** - Collapses or expands "Top level" group categories in the *Mesh Point Editor* all at once. Can be used to reduce the complexity of the table and focus on specific *Mesh Points*.

**Filter Rows** - Allows you to filter rows by using options to search for Values or Text in *Any* or *All* of the selected columns. Displays the *Mesh Point Filter* dialog box:

.




When filtering with *Value*, only display rows having values *Above* the *Maximum* value, *Below* the *Minimum* value, *Between* the range of *Maximum*/*Minimum* values, or *Outside* the range, in *Any* or *All* selected columns.

When filtering with *Text*, choose from *Contains* (only display rows when a portion of the text in *Any* or *All* selected columns contain all of the entered characters), *Not Contains* (only display rows when no portion of the text in *Any* or *All* selected columns contain all of the entered characters), *Equals* (only display rows when all text in *Any* or *All* selected columns exactly matches all of the entered characters), or *Not Equals* (only display rows when all text in *Any* or *All* selected columns does not exactly match all of the entered characters). All options are not case-sensitive.

Along with checking and unchecking the boxes, you can also highlight the *Columns to Check* one at a time, then click the *Toggle Selected On* or *Toggle Selected Off* icon buttons. Once all desired columns are selected, click OK and view the modified *Mesh Point Editor*.

**Select Entities to Filter Rows** - Allows you to select Mesh Points using the standard entity selection dialog, then removes the selected Mesh Points from the *Mesh Point Editor*.

**Clear Filter** - Clears all filter settings and all Mesh Points previously shown in the *Mesh Point Editor* before filtering will be visible again.

**Update Selection to Visible Rows** - This icon will update the selection list created by the *Select* toolbar with only the visible rows in the *Mesh Point Editor*. This command can be used when the filter is on or off, as well as, when there are groups in the *Mesh Point Editor*. This command will completely purge the selection list of all other entity types other than the underlying geometric points of the Mesh Points in the *Mesh Point Editor*, so be careful about using it if you have created a large selection list with multiple entity types.

**Show Visible Rows** - Highlights (using the options currently set in the *Window, Show Entities* command) all Mesh Points that are currently visible as rows in the *Mesh Point Editor*.

**Show When Selected Menu** - Contains a number of options to “show” the Mesh Points currently highlighted in the *Mesh Point Editor* by “highlighting/showing” them in the main graphics window. By default, this command is set to *Off*. The commands on this menu use the options found in the *Window, Show Entities* command (see Section 6.3.2.3, "Window, Show Entities..."). Once the options have been selected, simply clicking the *Show When Selected* icon will toggle this mode on and off.

Setting the menu to *Highlight* will highlight the selected Mesh Points 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 Mesh Points NOT associated with the selected entities in the *Mesh Point Editor* 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* (not used when highlighting Mesh Points) options can be toggled on and off and the highlight color can also be chosen from the FEMAP color palette by using the *Highlight Color...*option.

### Other Mesh Point Editor Features

**Expand/Collapse** toggles - Only appear when using “Groups” in the *Mesh Point Editor*. Allow you to collapse and expand individual group categories and sub-categories inside the *Mesh Point Editor*. When the toggle shows a “+” it means the group category can be expanded, when it shows a “-” it means the group category can be collapsed.

**Context Sensitive Menu (Rows)** - When a row or rows are highlighted, right mouse clicking will bring up a context sensitive menu. The commands on this menu allow you to do a number of different things:

***Show Mesh Point (# Selected)*** - “Shows” (highlights in the graphics window) the Mesh Point(s) corresponding to the row(s) currently highlighted in the *Mesh Point Editor*.

***Assign Mesh Point*** - When the Mesh Point(s) corresponding to the row(s) currently highlighted in the *Mesh Point Editor* are within the tolerance specified in the *Geometry Connections* section of the *Mesh Point Options* dialog box, they will be automatically associated to the closest piece of geometry.

***Delete Mesh Point Definitions*** - Deletes the Mesh Point Definitions associated with the Mesh Point(s) corresponding to the row(s) currently highlighted in the *Mesh Point Editor* and sets *Definition* column to “0,,None”.

***Delete Mesh Points*** - Deletes the Mesh Point(s) corresponding to the row(s) currently highlighted in the *Mesh Point Editor*.

***Select Mesh Points to Delete*** - Select Mesh Points to delete from the *Mesh Point Editor* using the standard entity selection dialog box.

***Select Mesh Points to Toggle On/Off*** - Select Mesh Points to toggle the current state of the check box in the *On/Off* column of the *Mesh Point Editor* using the standard entity selection dialog box.

***Select Mesh Points to Toggle Locked*** - Select Mesh Points to toggle the current state of the check box in the *Locked* column of the *Mesh Point Editor* using the standard entity selection dialog box.

***Select All*** - Selects all rows and columns currently in the Mesh Point Editor.

***Copy Rows, Copy Rows - No Headers, Save Rows,*** **and** ***Filter Rows*** - Highlighted rows can be copied to the clipboard using *Copy Rows*, copied to the clipboard without column header information using *Copy Rows - No Headers*, saved to a text file using *Save Rows*, or placed into a “filter” using *Filter Rows*.

***Note:*** It is possible to choose multiple entity rows at the same time and perform operations on all of the rows at once. In order to choose multiple entity rows, hold down the CTRL key and select individual entities with the mouse or hold down the SHIFT key and select a first row and a last row and all of the rows in between will also be selected.

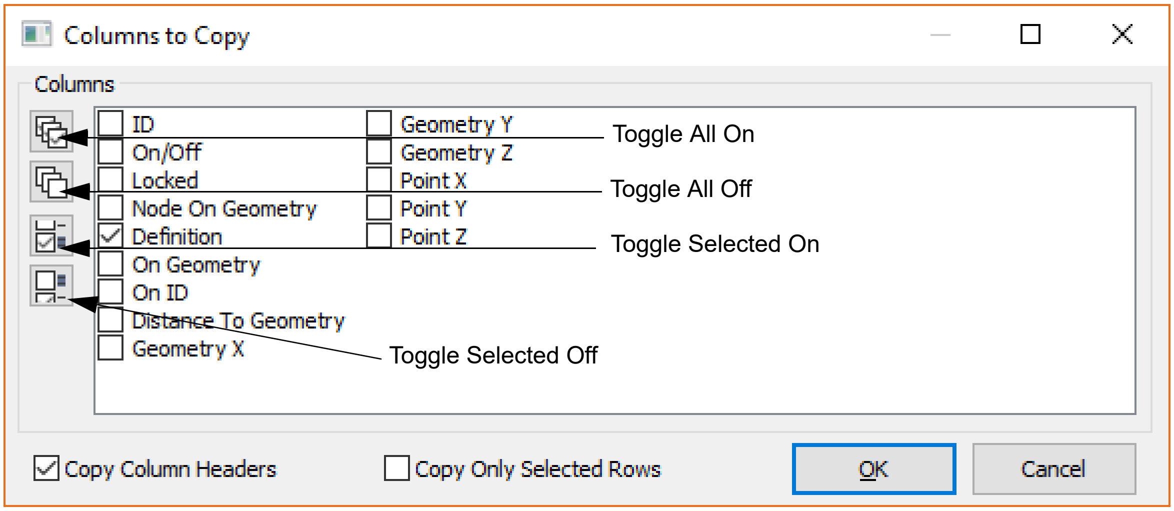
**Context Sensitive Menu (Column Header)** - When the cursor is positioned over a column header, right mouse clicking will bring up a context sensitive menu. This menu allows you to:

***Sort Ascending*** **or** ***Sort Descending*** - uses the values under the selected column header.

***Sort Off*** - use *Sort Off* to return the *Connection Editor* to a state before any sorting was done.

***Copy Columns*** - brings up the following dialog box. Check the desired columns to copy to the clipboard and by default those columns will be copied in their entirety. There are also options to *Copy Column Headers*, which is on by default, which will include the title of the column, and *Copy Only Selected Rows* which will send only the rows currently highlighted in *Connection Editor* and located in a “checked” column to the clipboard

***Note:*** The combination of copying columns with the *Copy Only Selected Rows* option turned on is a convenient method to transfer only the necessary data to your exterior spreadsheet program.




***Group by Column*** - adds a copy of the column header to the *Mesh Point Editor* "Group Header" section.

***Hide Column*** - quickly hides the specified column. Any column can be shown again using the *Show/Hide Columns* icon menu in the *Mesh Point Editor*.

## Model Info tree

- Added *Connector Set* command to context-sensitive menu for Connectors, which allows creation of a Connector Set containing any number of Connectors.
- Added *Text* entities to the *Selection List* when they are the active entity in the *Select* toolbar. Also, implemented context-sensitive menu for text items in the *Selection List*.

## Meshing Toolbox

- Added *Slice* to the *Operation* section of the *Geometry Editing* tool, which is used to divide surfaces, including boundary surfaces, using a slice plane defined after the surfaces have been selected.
- Added *Midpoint* and *Curve/Location* options to the *Break Method* section when *Operation* is set to *Curve Break* in the *Geometry Editing* tool. *Midpoint* breaks the selected curve(s) at the midpoint of each curve, while *Curve/Location* breaks the selected curve(s) using the closest distance between the specified location and each curve.
- Added *Include Internal Loops* and *Split Solid* options when *Operation* is set to *Pad* in the *Geometry Editing* tool. The *Split Solid* option attempts to subdivide solid geometry using solids generated from the "pad" or "pad with washer" pattern.
- Added *Include Non-Circular Loops*, *Include Non-Circular Loops*, and *Split Solid* options when *Operation* is set to *Washer* in the *Geometry Editing* tool. The *Split Solid* option attempts to subdivide solid geometry using solids generated from the "washer" pattern.
- Added *Along Vector* option to *Project/Move To* section when *Operation* is set to *Project/Move Point* in the *Geometry Editing* tool. This option moves/projects the selected point(s) along a vector specified after the points have been selected. This can be useful when trying to modify the overall shape of solids and surfaces.
- Added ability to use the "Esc" key to deactivate the *Select* icon for all tools which feature entity selection.
- Improved performance when automatically updating rigid elements as geometry is being remeshed via the Meshing Toolbox. For example, as test model with approximately 1.4M Nodes and 1.1M Elements, including 800 Rigid elements, demonstrates an 8X performance improvement for each individual mesh size change.

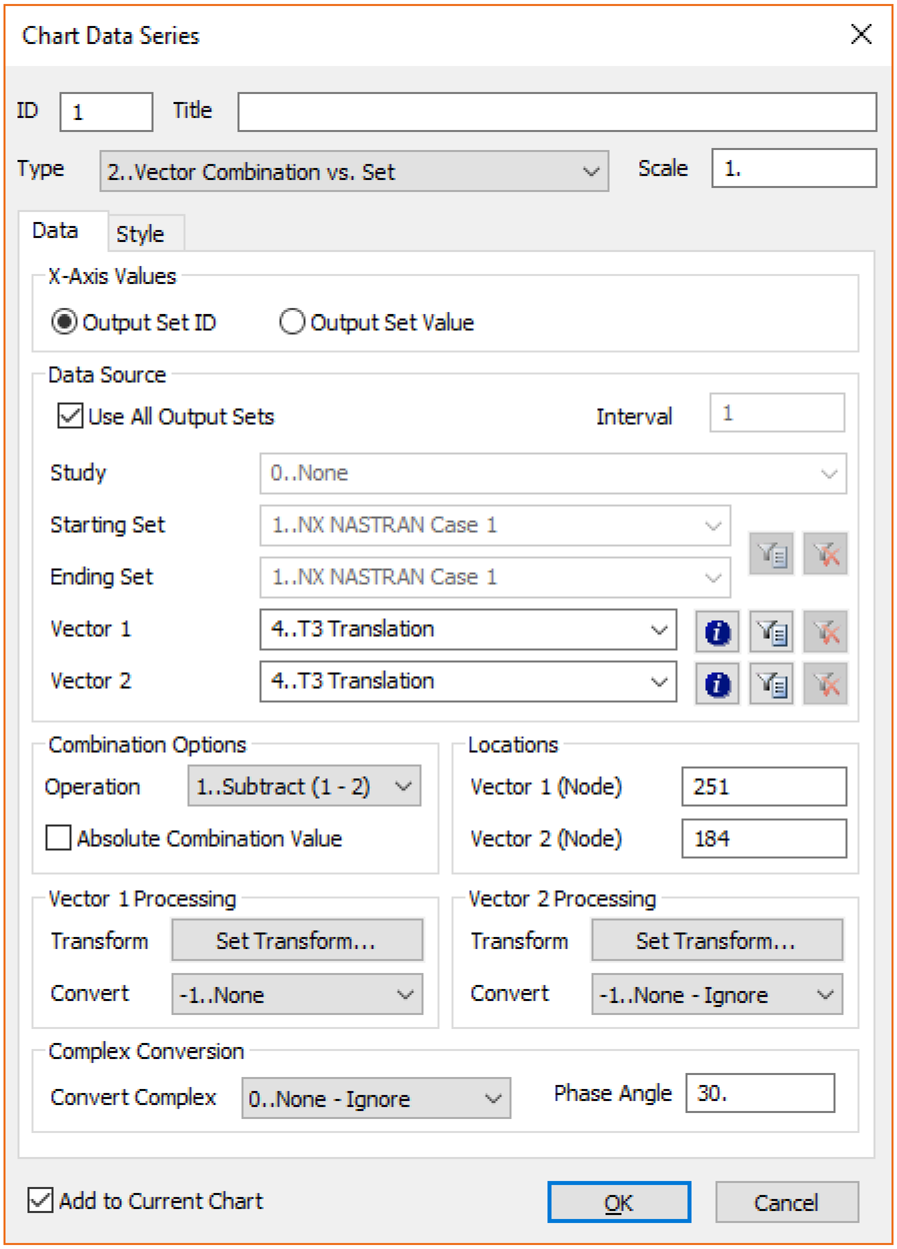
## PostProcessing Toolbox

- Added *Dynamic Control* button to the *Criteria* section, which accesses the *View, Advanced Post, Dynamic Criteria* command. See Output and Post-Processing for more information.

### Charting pane

- Improved algorithm in FEMAP 12.0.1 for plotting angular values in "0..Vector vs Entity" Chart Data Series, set to *Position*, when transforming to cylindrical or spherical coordinate systems. Rather than only transforming polar to Cartesian, values are also offset by 360 degrees, as appropriate, to create most continuous plot possible.
- Added "2..Vector Combination vs. Set" option to the *Type* drop-down in the *Chart Data Series* dialog box.

Plots XY data versus the *Output Set ID* or *Output Set Value* for a "combination" of *Output Vectors* across several *Output Sets*.




The *X-Axis Values* section offers two options, *Output Set ID* or *Output Set Value*. When *Output Set ID* is selected, the IDs of the specified output sets will be used as the X values, while *Output Set Values* will use the values of the specified output sets as the X values. The *Data Source* section is used to specify a range of output sets and two output vectors, specified using the *Vector 1* and *Vector 2* drop-downs, which will be combined using one of the operations available in the *Combination Options* section for the data series. *Vector 1* and *Vector 2* can be the same. By default, the *Use All Output Sets* option will be "on", which will simply use all available output sets in the model. If *Use All Output Sets* option is "off", a range of output sets can be specified by selecting an Analysis Study from the *Study* drop-down or by using the *Starting Set* and *Ending Set* drop-downs. Optionally, an *Interval* value can be specified, so only every n-th output set in the range will be used.

The *Locations* section is used to specify an entity for *Vector 1* and *Vector 2*, which must be a node or element ID in the model. A node ID must be specified when a nodal output vector is selected, while an element ID must be specified for an elemental output vector. *Vector 1* and *Vector 2* can be the same. Typically, the type of output should be the same, but it is not required.

The *Combination Options* section contains the *Operations* drop-down, which contains 4 options which can be used to combine data from *Vector 1* with data from *Vector 2*. Choose an operation from “0..Add (1+2)”, “1..Subtract (1-2)”, “2..Multiple (1*2)”, or “3..Divide (1/2)”. When *Absolute Combination Value* is enabled, the absolute value of the specified combination will be used. If using “3..Divide (1/2)” and any values in *Vector 2* are zero, an error will be issued stating the data series is possibly invalid.

The settings for the *Transform* and *Convert* options in the *Vector 1 Processing* and *Vector 2 Processing* sections, which can be different from one another, along with the *Convert Complex* option in the *Complex Conversion* section, are described above in Output Processing.

### Data Surface Editor

- Added *Criteria* option to *Output Map Data Conversion Options* section of *Define Options for Variation* dialog box for use with the *Output Map Data Surface*. Also, added ability to use *Criteria* option to create Nodal Loads for FEMAP 12.0.1.
- Added *Preview* icon button to *Define Variation Between Coordinates Data Surface* dialog box, which graphically displays lines between 2, 4 or 8 locations. This can be helpful when trying to determine if any “twisting” has occurred during selection of coordinates, especially when using a 4-point or 8-point data surface. In addition, added checking which alerts the user if FEMAP believes the data surface is “twisted”.

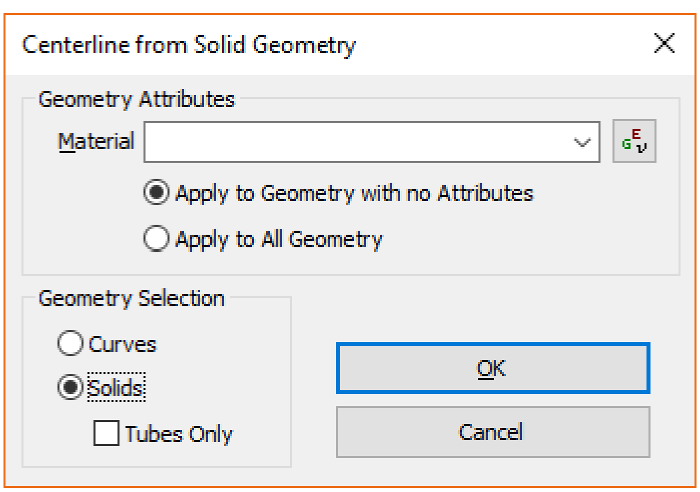
### Data Table

- Added *Transform* button to *Results to Add to Data Table* dialog box, which is accessed via the *Add Output Columns* icon. Clicking this button displays the standard *Results Transformation* dialog box, which is used to select transformation options for Nodal Vector Output being added when nodes are already in the Data Table or Element Output on Line Elements, Shell Elements, and/or Solid Elements already in the Data Table.
- Added *Filter Selected Entities* button to *Data Table Filter* dialog box, which is accessed via the *Filter Rows* icon. Clicking this button displays the standard entity selection dialog box for the type of entity which is currently in the *Data Table* and the rows representing the selected entities are removed from the *Data Table*.

## Geometry

- Added *Modify, Reflect* menu which can be used to reflect Points, Curves, Surfaces, or Solids without creating a copy. When reflecting a geometric entity, all geometric entities needed by the entity will also be reflected.
- Added *Geometry, Curve, Centerline* command, which attempts to create a curve which represents the centerline of a geometric solid.

This command can behave somewhat differently, depending on the options specified in the *Centerline from Solid Geometry* dialog box:



The type of geometry selected by the command, solids or a curve on a solid, specifies which approach will be used to determine and create the centerline curve. In addition, an attempt will be made to create a new beam property which represents the cross-section of each selected solid, then use the new property as a meshing attribute on each newly created centerline curve. See Section 5.1.2.11, "Mesh, Mesh Control, Attributes Along Curve".

### Geometry Attributes

This section specifies which material(s) will be referenced by properties automatically created by this command. Use the *Material* drop-down to select an existing material or use *New Material* icon button to create a new material.

***Apply to Geometry with no Attributes*** - when this option is selected, properties created by this command will use the material selected in the *Material* drop-down, unless various property meshing attributes were already assigned to the original solids. In that case, the material referenced by the property meshing attribute on each solid will be used by the corresponding new property created for each solid.

***Apply to All Geometry*** - when this option is selected, properties created by this command will all use the material selected in the *Material* drop-down, regardless of property meshing attributes already assigned to original solids.

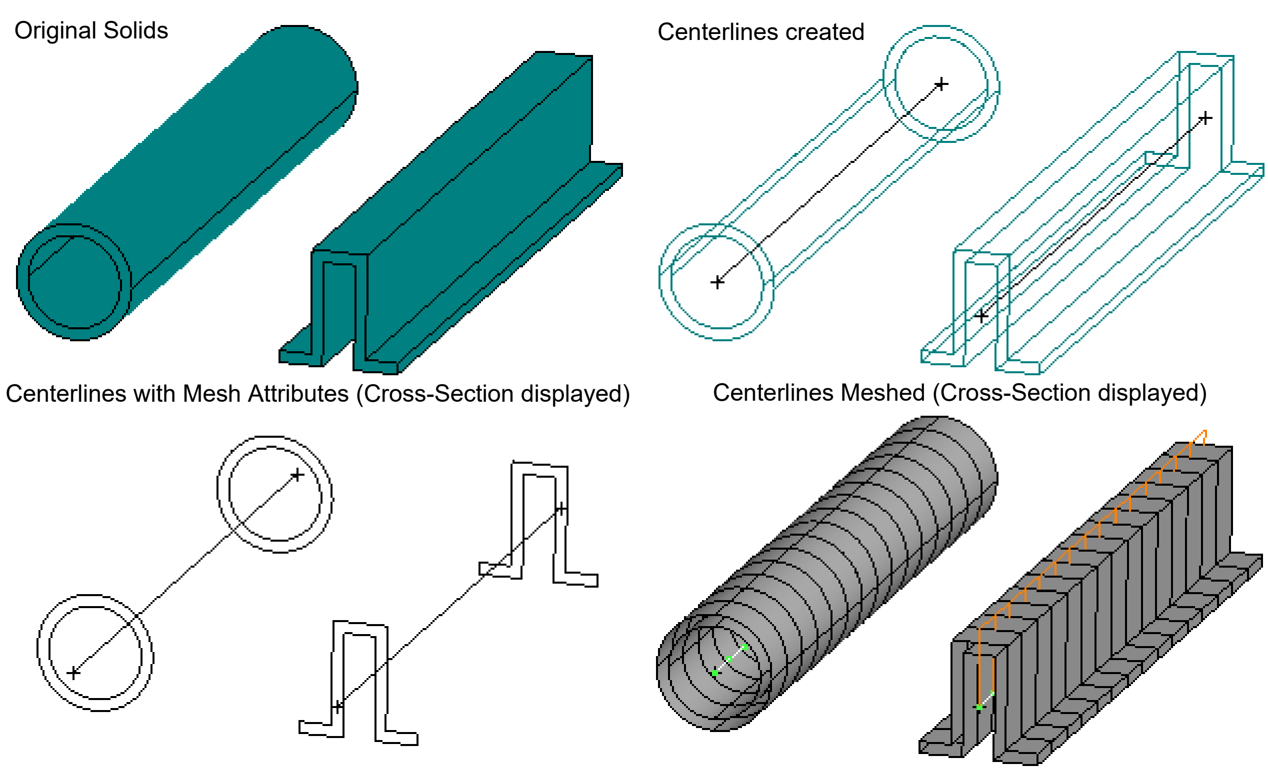
### Geometry Selection

This section specifies which type of geometric entity will be selected by the command. The approach used to determine the appropriate location to create the centerline curve is different depending on this selection.

***Curves*** - when using this option, only a single curve on a solid can be selected. After the curve is selected, the mid-point of the curve is calculated and a cross-section is determined by slicing the solid at that location. This cross-section is then used to create the beam property which will be assigned to the centerline as a meshing attribute.

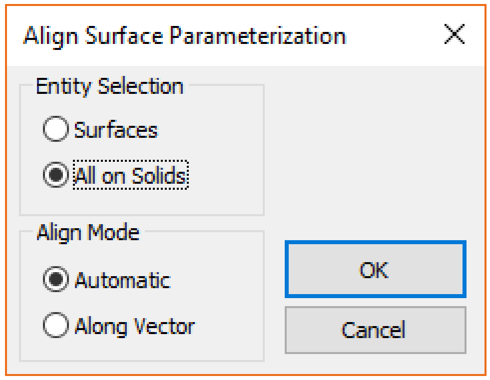
***Solids*** - when using this option, any number of solids can be selected at one time. Once selected, the centerline will be determined by one of two approaches. The first approach searches for cylindrical and toroidal surfaces, which, when found, are used to determine the cross-section for the property to assign to the curve as a meshing attribute. If no circular cross-section can be found, the second approach is to use the method described above for *Curves* on the longest edge of the solid. If *Tubes Only* option is enabled, only the first approach is used.

***Note:*** Any circular cross-sections found using the various approaches described above will create a beam property with *Shape* set to “Circular Bar” or “Circular Tube”. All other cross-sections will have *Shape* set to “General Section”, regardless if the shape matches a “standard” section.



- Added *Modify, Update Other, Align Surface Parameters* command.

This command attempts to modify surface parameters so all “periodic splits” in cylindrical or conical geometry are “aligned” with one another as much as possible. The first step is to select options in the *Align Surface Parameterization* dialog box:



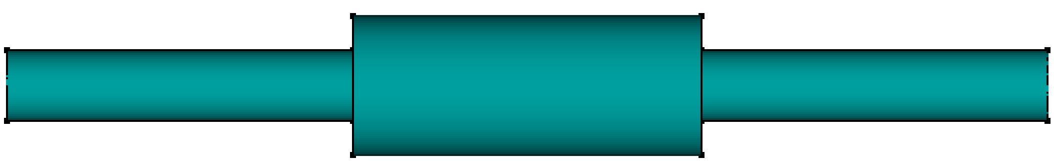

The *Entity Selection* section allows selection, via the standard entity selection dialog box, of *Surfaces* (considers only selected surfaces) or *All on Solids* (considers all surfaces on selected solids).

The *Align Mode* sections offers two options, *Automatic* or *Along Vector*. When using the *Automatic* option, the command will simply pick a “zero” location for the periodic split of the cylindrical or conical geometry, then attempt to rotate various surfaces to align all of the periodic splits. When using *Along Vector* option, a vector specified using the standard vector definition dialog box will be used to specify the “zero” location (base of vector) and the location 180 degrees on the other side of the geometry (tip of vector) to be used to align the surfaces.

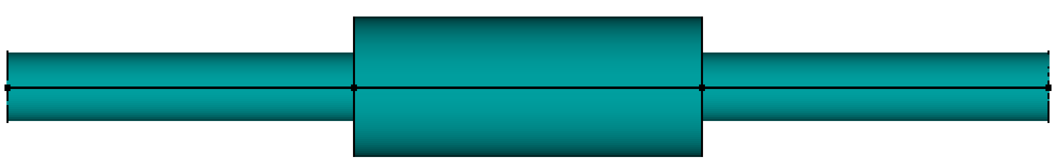
For Example:

Original Geometry - Notice splits in different portions of the solid are not aligned

Using All on Solids and Automatic options, all splits aligned, surfaces rotated 90 degrees



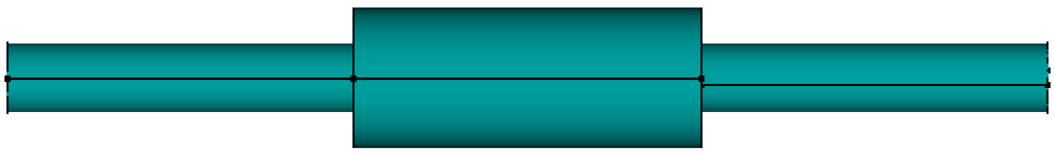
Using All on Solids and Along Vector options, vector defined normal to screen



Using Surfaces option (selected Surfaces highlighted)...

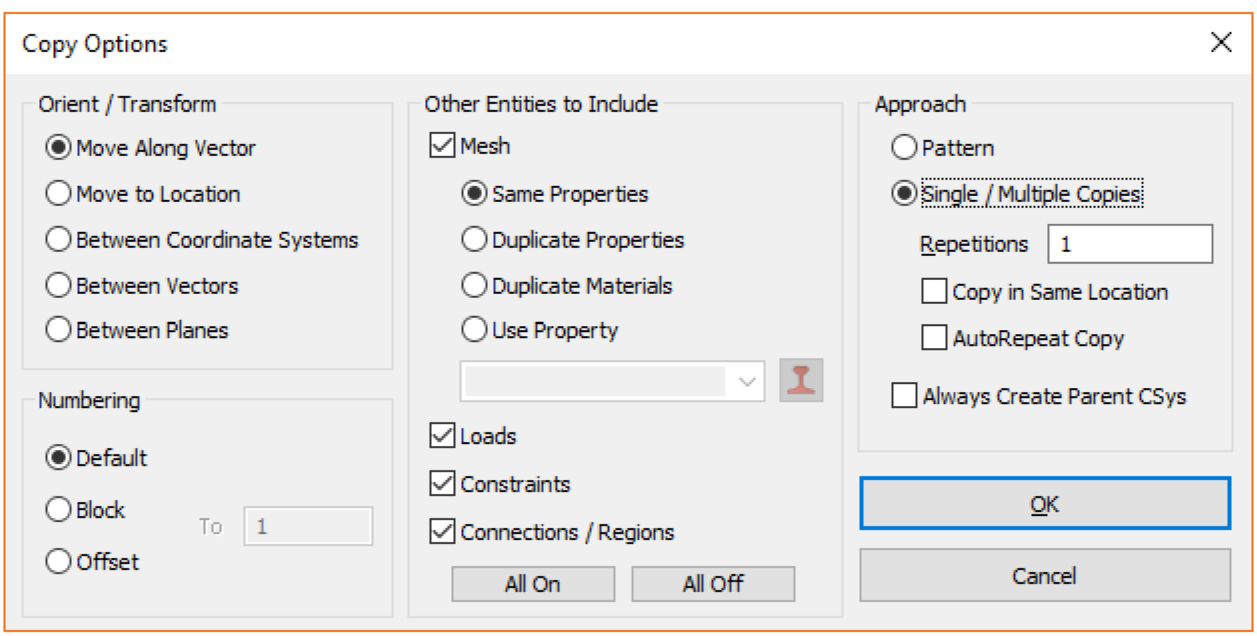
Using Automatic option

Using Along Vector option, vector defined normal to screen



- Enhanced several commands which perform Boolean and other operations on solid geometry to work properly with geometric entities that contain boundary surfaces and/or combined curves.
- Enhanced *Geometry, Solid, Stitch* command to use the entered *Gap Tolerance* as a the largest tolerance to use, as tolerance is now increased incrementally as the geometry is checked for connectivity after each step.
- Enhanced the *Geometry, Surfaces, Ruled* command to be *Geometry, Surfaces, Ruled / Between Curves*, which adds options to control surface tangents on one or both sides of the newly created surface.
- Enhanced commands for geometric entities on the *Modify, Move By* menu, the *Modify, Rotate By* menu, and the *Modify, Align* menu by adding an option to also move/rotate/align mesh associated with selected geometric entities. In addition, added *Always Create Parent CSys* option, which can be used to force creation of a "parent" coordinate system to be used as the reference coordinate system for any coordinate systems which need to be created to maintain the proper definition of finite element entities, such as boundary conditions or regions.
- Enhanced the commands on the *Geometry, Copy* menu, the commands *Geometry, Rotate* menu, and the commands on the *Geometry, Reflect* menu by adding an option to automatically select associated mesh, along with additional options.

You can use the Copy/Rotate/Reflect commands to duplicate existing points, curves, surfaces, volumes, or solids. All *Geometry, Copy* commands require the exact same input, independent of the geometry being copied. After selecting the appropriate command for the type of entity to select, the standard entity selection dialog box will appear. Here, the entities are copied using the options specified in the *Orient/Transform* section of the *Copy Options* dialog box:



Each section of the *Copy Options* dialog box offers a number of different options, most of which can be used in conjunction with one another, to perform complex operations which can potentially provide a streamlined workflow when copying entities. In addition, it offers tremendous flexibility in terms of what other entity types to include when creating copies of the entity type selected during the chosen command.

### *Orient/Transform - Geometry, Copy commands*

This section offers five different options to orient and/or transform copied finite element entities:

***Move Along Vector*** - Copied entities are translated along a vector specified using the standard vector definition dialog box.

***Move to Location*** - Copied entities are translated along a vector specified using standard coordinate definition dialog boxes to first specify a "Copy From" location, then a "Copy To" location. This may be useful if using different *Methods^* for selected coordinates to use as the "Copy From" and "Copy To" locations.

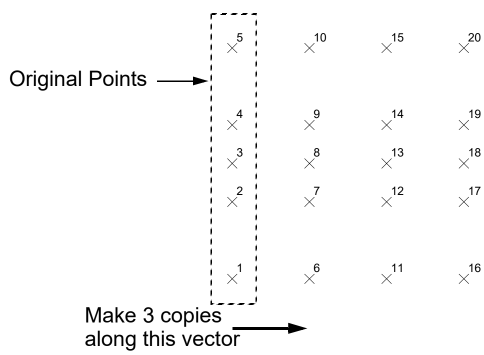
***Between Vectors*** - Copied entities are "aligned" using two vectors, each specified using a standard vector definition dialog box. The first vector defines the original position and orientation that will be aligned. The second vector defines the new or desired position and orientation. The entities are first translated from the origin of the first vector to the origin of the second vector, then rotated based on the angle between the vectors.

***Between Planes*** - Copied entities are "aligned" using two planes, each specified using a standard plane definition dialog box. The behavior is similar to *Between Coordinate Systems*, only each specified plane (X and Y axes) and each plane's normal direction (Z axis) are used to determine XYZ axes to align.

***Between Coordinate Systems*** - Copied entities are "aligned" using two coordinate systems selected using the *From CSys* and *To CSys* drop-downs in the *Copy Between Coordinate Systems* dialog box. The XYZ axes of the *From CSys* coordinate system will be aligned to the XYZ axes of the *To CSys* coordinate system and copies of all the selected and associated entities will move as a rigid body to the newly aligned location. The type of coordinate system used for *From CSys* and *To CSys* does not matter, as only the axes are aligned.

***Note:*** If using a cylindrical coordinate system, the R (radial) axis is the X axis, the T (theta) axis corresponds is the Y axis, and the Z axis is the Z axis. If using a spherical coordinate system, the R (radial) axis is the X axis, the P (phi) axis is the Y axis, and the T (theta) axis is the Z axis.

This figure shows the original nodes copied three times using the *Move Along Vector* option in *Orient/Transform* and *Repetitions* in the *Approach* section set to "3".




### Copying in Non Rectangular Coordinates

When using the *Move Along Vector* or *Move to Location* options, copies are always created along the specified vector, that is along a straight line. The vector can be specified in any convenient coordinate system. The *Geometry, Copy* command cannot be used to create a copy in a rotated location by choosing the angular direction in a cylindrical coordinate system. If this is desired, use the *Geometry, Rotate* commands to create rotated copies.

### *Orient/Transform - Geometry, Rotate commands*

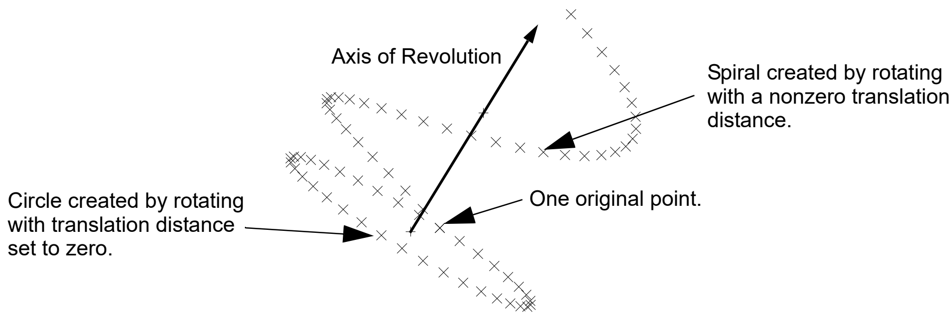
This section offers two different options to orient and/or transform copied finite element entities:

***Rotate Around Vector*** - Copies of entities are rotated around a vector specified using the standard vector definition dialog box, using values specified for *Rotation Angle* and *Translation Distance* specified in the *Change per Repetition* dialog box. Each copy is rotated around the axis of rotation vector by the specified angle (following right-hand rule conventions), and is translated along the axis vector by specified distance. If a non-zero value for *Translation Distance* is specified, a spiral will be created.

***Rotate to Location*** - Copies of entities are rotated around a vector specified using the standard vector definition dialog box, using two locations, a “starting location” and “ending location”, specified using standard coordinate definition dialog boxes. This option may be useful if using different *Methods*^ for “starting location” and “ending location” or the *Rotation Angle* and/or *Translation Distance* are not known before starting the command.

For all of these commands, first, use the standard entity selection dialog box to select which entities to copy, then specify options in the *Copy Options* dialog box, then follow appropriate steps to complete the rotate operation.

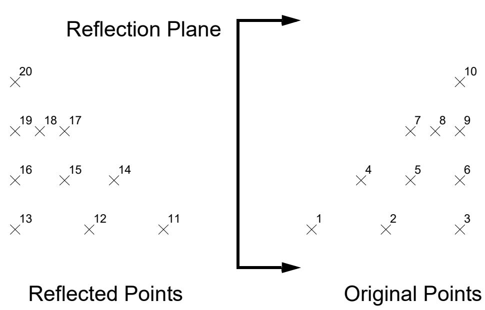
This figure show the same original point rotated using the *Rotate Around Vector* option with and without a value for *Translation Distance* specified:




### *Reflection Options - Geometry, Reflect commands*

This section offers only one option, *Trap Width*. A copy of any selected entity will not be made if it is closer to the reflection plane than the specified *Trap Width*. If *Trap Width* is set to zero, all of the selected entities will be reflected. This option is used most often when reflecting elements (to join nodes on the reflection plane), and will usually be zero when reflecting nodes.

As always, you will use the standard entity selection dialog box to choose the entities to be reflected. Then FEMAP will display the *Generation Options* dialog box. (See Section 3.5.1, "Geometry, Copy Commands"). All options work just like they do for *Geometry, Copy* commands, except that you cannot choose multiple repetitions.




Next, FEMAP will display the standard plane selection dialog box, so you can define the reflection plane. You can choose any plane that you want. It does not matter how your selected entities are oriented with respect to the plane. They can be on one side, or they can be on both sides of the plane. Just remember that the reflected entities will be located on the opposite side of the plane from the original.

### *Numbering*

This section offers three different options to control IDs of the duplicate entities:

***Default*** - Duplicated entities are numbered sequentially, based on original ID, and given the next available ID for each entity type.

***Block*** - Duplicated entities will have the ID of the duplicated entity with the lowest original ID, for each entity type, set to the value specified in the *To* field. Then, the constant difference between the lowest original ID and the value in the *To* field will be added to all other IDs. No sorting will take place, and no gaps will be filled. This is an easy method to maintain a numbering structure while shifting it to another level (such as making sure the copies of entities 1, 20, 50, 100, and 500 will be 1001, 1020, 1050, 1100, and 1500 by using a *To* value of 1001).

***Offset*** - Duplicated entities will be offset from the original ID of each entity using the value specified in the *By* field. No sorting will take place, and no gaps will be filled. This is an easy method to maintain a numbering structure while shifting it to another level (such as making sure the copies of entities 1, 20, 50, 100, and 500 will be 1001, 1020, 1050, 1100, and 1500 by using a *By* value of 1000).

### *Other Entities to Include*

This section contains four categories of entity types which can be included when copying geometry. The *All On* and *All Off* buttons can be used to quickly enable or disable all the currently available categories.

***Mesh*** - When enabled, all nodes and elements which are associated to the geometric entities being duplicated will also be duplicated. See *Property/Material Options* below for more information on the options in this sub-section.

***Loads*** - When enabled, any loads on geometric entities will also be duplicated. If the *Mesh* option is also enabled, any nodal and elemental loads on nodes and elements being duplicated will also be duplicated.

***Constraints*** - When enabled, any constraints on geometric entities being duplicated will also be duplicated. If the *Mesh* option is also enabled, nodal constraints on any duplicated nodes will also be duplicated. If all nodes used by a constraint equation are being duplicated, then the constraint equation will also be duplicated.

***Connections/Regions*** - When enabled, any full or partial regions defined by geometric entities being duplicated will also be duplicated. This is the case for Connection, Fluid, NonStructural Mass, Bolt, and Rotor regions. If the *Mesh* option is also enabled, any full or partial regions defined by nodes or elements being duplicated will also be duplicated. In addition, if both Connection regions of a Connector are copied, then the Connector is also duplicated.

### *Property/Material Options*

This sub-section offers four different options for defining properties and materials for duplicated entities:

***Same Properties*** - Duplicated elements simply use the same property currently referenced by the original elements. When the *Geometry* option is enabled, any property meshing attribute assigned to the original geometric entity will also be assigned to the duplicated geometric entity.

***Duplicate Properties*** - Duplicated elements will reference a duplicate of the property currently referenced by the original elements. When the *Geometry* option is enabled, any property used as a meshing attribute assigned to the original geometric entity will duplicated, then the duplicate will be assigned to the duplicated geometric entity.

***Duplicate Materials*** - Same as the *Duplicate Properties* option, only each duplicate property will reference a duplicate of the material referenced by the original property. When the *Geometry* option is enabled, same as the *Duplicate Properties* option, only each duplicate property created for use as a meshing attribute will reference a duplicate of the material referenced by the original property.

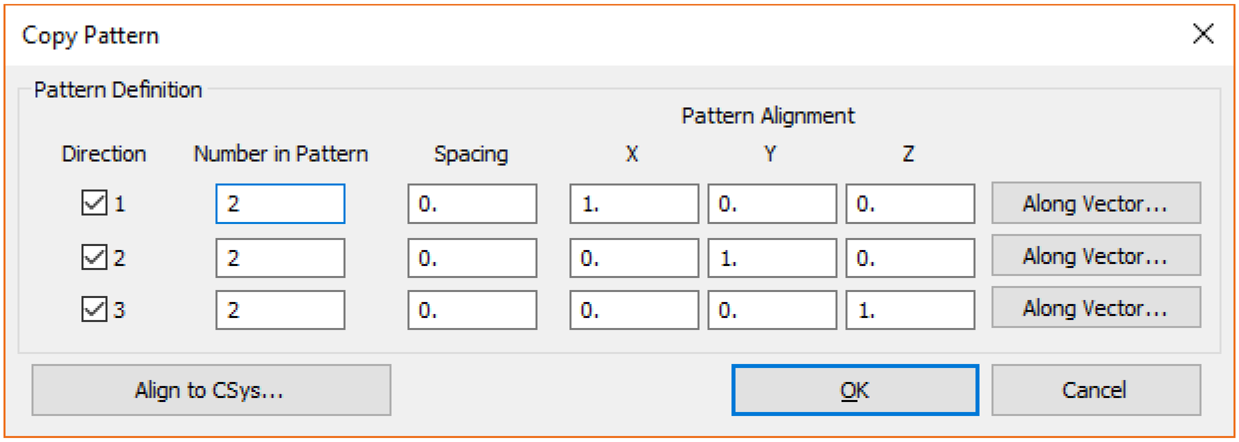
***Use Property*** - Attempts to change the property of duplicated elements to the property selected in the drop-down. Will only change the property if the duplicated element is the appropriate type to reference the selected property. When the *Geometry* option is enabled, attempts to assign the selected property as a meshing attribute to any geometric entity which had a property meshing attribute specified before being duplicated. Again, property will only be updated if it matches the property type assigned as a meshing attribute to original geometric entities.

### *Approach*

This section offers a choice between creating a *Pattern* of duplicate entities or *Single/Multiple Copies* of the selected entities. While *Pattern* may be useful in certain situations, *Single/Multiple Copies* offers some additional options, which allow for a more generalized and flexible approach when creating duplicates.

### Pattern - Copy commands only

When using *Pattern*, the *Copy Pattern* dialog box will be shown after clicking OK to the *Copy Options* dialog box:




The *Copy Pattern* dialog box is setup to create a copy of the originally selected entities using up to three different directions, in series. The values for *Number in Pattern* represents the "total" number of entities which will be created in each direction, while the value for *Spacing* represents the distance between each duplicate in each direction of the pattern. Duplicates of the selected entities are first created in *Direction 1*, then duplicates of the original entities and the duplicate entities created in *Direction 1* are created in *Direction 2*, then duplicates of the original entities and the duplicate entities created in *Direction 1* and *Direction 2* are created in *Direction 3*.

The *Along Vector...* buttons may be used to define a vector for any or all directions using the standard entity selection dialog box, while the *Align to CSys...* button will transform the values currently specified in the *Pattern Alignment* section to represent unit vector values in the selected coordinate system.

For example, imagine copying a single selected entity with *Number in Pattern* for *Direction 1* is set to "2", *Direction 2* is set to "3", and *Direction 3* is set to "4". In this case, a single copy of the selected entity is created in *Direction 1* (total entities now 2), two copies of each of those 2 entities are created in *Direction 2* (total entities now 6), then three copies of each of those 6 entities are created in *Direction 3* (total entities at end of command, 24).

### Single/Multiple Copies

When using *Single/Multiple Copies*, a number of additional options are available.

***Repetitions*** - By default, this value is set to "1", which creates a single copy of each selected entity. To create multiple copies of each selected entity, simply set this option to the number desired. While not required, this value should typically be set to "1" when using the *AutoRepeat Copy* option, as the command will then create a single copy of each selected entity at each new location.

***Copy in Same Location*** - When enabled, creates a copy of the selected entities without specifying any additional information for the options in the *Orient/Transform* section (i.e., creates coincident entities).

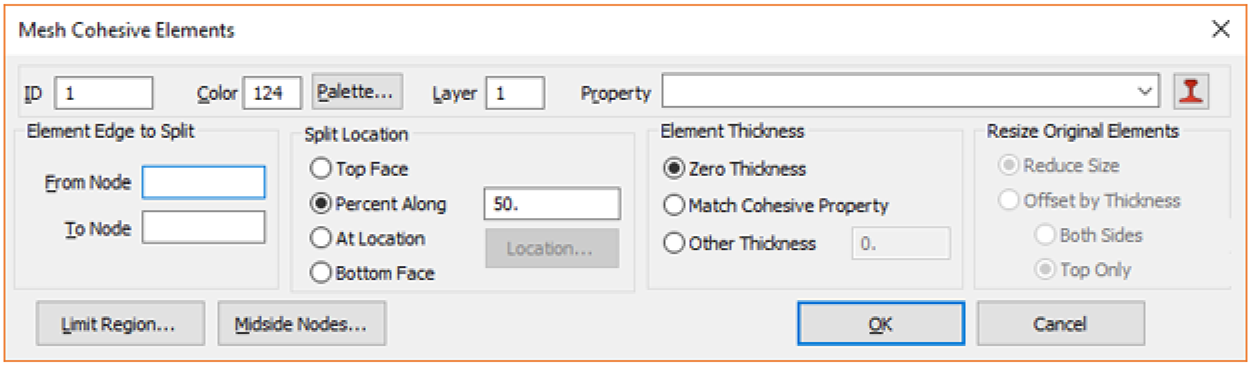
***AutoRepeat Copy*** - When enabled, repeats all dialog boxes required by the currently selected option in the *Orient/Transform* section after saying OK to the *Copy Options* dialog box (i.e., selected entities and options specified in *Copy Options* are used for all subsequent copy operations). The dialog boxes will continue to repeat until *Cancel* is pressed, thus ending the command. To avoid multiple copies of the selected entities being created at each new location, set *Repetitions* to "1".

***Always Create Parent CSys*** - When enable, forces the creation of a "parent" coordinate system, which is used as the reference coordinate system for any coordinate systems which need to be created to maintain the proper definition of finite element entities, such as boundary conditions or regions.

## Meshing

- Added *Modify, Reflect* menu which can be used to reflect Nodes or Elements without creating a copy.
- Added the *Mesh, Editing, Cohesive Meshing* command to create a layer of Cohesive Elements.

This command can be used to insert cohesive elements into an existing mesh. Similar to the *Mesh, Editing, Edge Split* command, everything is based on selection of 2 "corner" nodes on a single element edge (i.e., midside nodes cannot be selected). The "direction" of the split (i.e., "split path") is initially perpendicular to the selected element edge, then continues element-by-element until all appropriate elements have been split. This command offers some additional options, specific to the insertion of cohesive elements, which are controlled using the *Mesh Cohesive Elements* dialog box:




### Element Edge to Split

This section is simply used to select the *From Node* and the *To Node* to define the element edge to split.

### Split Location

This section has four options which specify the location of the split along the selected element edge to use for insertion of cohesive elements. "Top Face" and "Bottom Face" are determined by the selected *From Node* and *To Node*. The element face which includes the *To Node* and is opposite of the *From Node* is the "Top Face", while the element face which includes the *From Node* and is opposite of the *To Node* is the "Bottom Face".

***Top Face*** - inserts a layer of cohesive elements along the "top face" of the element whose nodes were selected in the *Element Edge to Split* section. Using this option does not "split" any existing elements, but instead separates the mesh above and below the "top face", then uses the nodes of the existing elements for the layer of cohesive elements, which reconnects the mesh.

***Percent Along*** - inserts a layer of cohesive elements between two sets of elements created by splitting a set of existing elements. Uses the nodes selected in the *Element Edge to Split* section and the value specified for *Percent Location* to determine the location to split all of the elements split by the command. The specified value is the percentage of the distance from the *From Node* to the *To Node* and this percentage is calculated for each element edge being split.

***At Location*** - inserts a layer of cohesive elements between two sets of elements created by splitting a set of existing elements. Uses the nodes selected in the *Element Edge to Split* section and the location specified using the *Location* button to determine the location to split the first element, then attempts to maintain a single distance from the *From Node* for all other elements split by the command.

***Bottom Face*** - inserts a layer of cohesive elements along the "bottom face" of the element whose nodes were selected in the *Element Edge to Split* section. Using this option does not "split" any existing elements, but instead separates the mesh above and below the "top face", then uses the nodes of the existing elements for the layer of cohesive elements, which reconnects the mesh.

### Element Thickness and Resize Original Elements

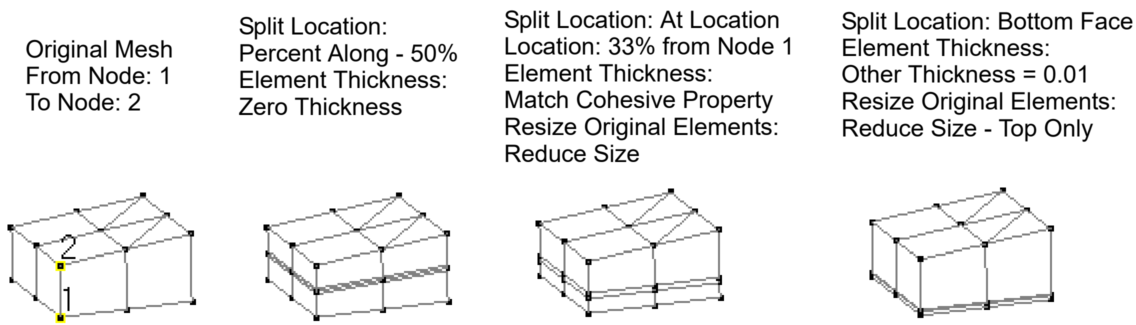
Unlike other solid elements, cohesive elements can have "zero physical thickness" (i.e., the nodes of the "bottom face" of wedge and hex elements which reference a Solid Cohesive property can be coincident with the corresponding nodes of the "top face"). This is allowed because the important stiffness values of a cohesive element can be explicitly specified by having the cohesive property reference a cohesive material (MATCZ). Cohesive properties can also reference an isotropic material (MAT1) or a 3-D orthotropic material (MAT11), where certain values from the material are combined with the thickness specified on the cohesive property to calculate the stiffness values. The physical thickness of a cohesive elements is never used to calculate the stiffness values.

The *Resize Original Elements* section contains options which are only available when *Element Thickness* is set to *Match Cohesive Property* or *Other Thickness* and will explained in the description for *Match Cohesive Property.*

***Zero Thickness*** - the layer of cohesive elements is created with zero physical thickness at the specified split location. The outer extents of each original element remain the same.

***Match Cohesive Property*** - since the layer of cohesive elements will have a physical thickness, in this case, the thickness specified on the selected solid cohesive property, any existing element being split will need to either be reduced in size or offset from the split location. When *Resize Original Elements* is set to *Reduce Size*, the elements on each side of the split will have their thickness reduced by 1/2 of the thickness specified on the cohesive property. When *Resize Original Elements* is set to *Offset by Thickness*, two choices are offered. When using *Both Sides*, the elements on each side of the split, along with any elements connected to those elements, will be offset 1/2 of the thickness specified on the cohesive property in both directions. When using *Top Only* only the elements on the “top half” of the split (i.e., the set of elements closer to the *To Node*), along with elements connected to those elements, will be offset the full thickness specified on the cohesive property in a single direction.

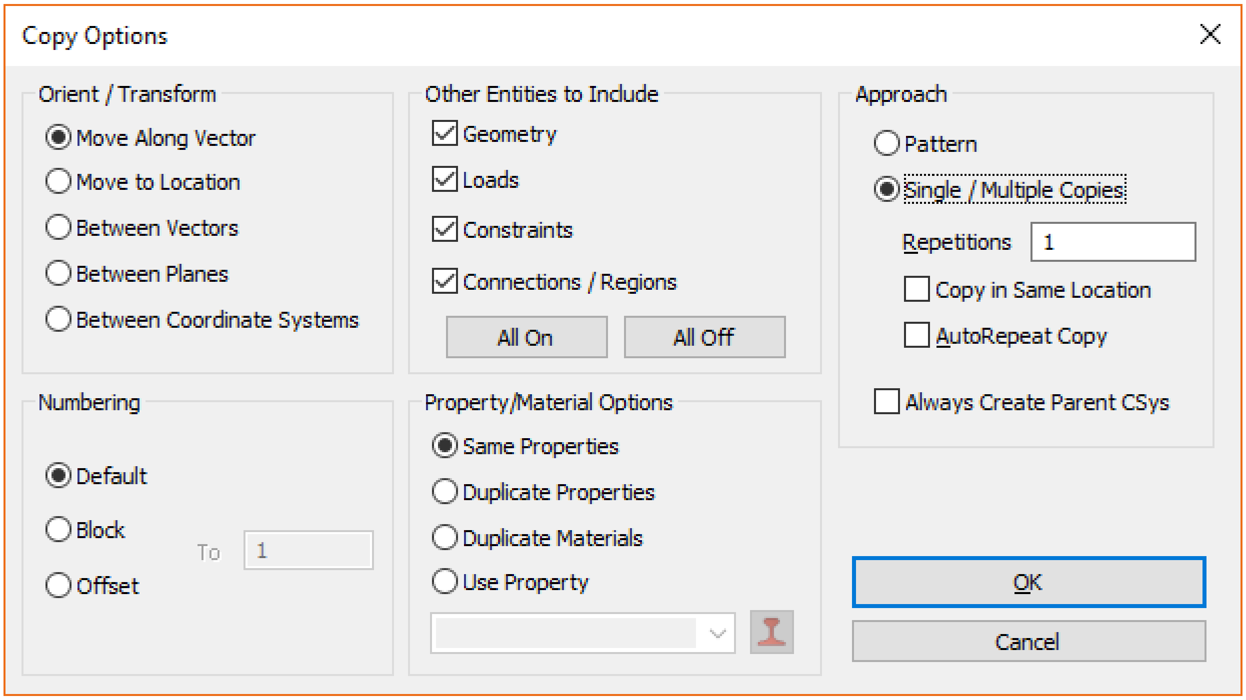
***Other Thickness*** - Behaves in the same manner as *Match Cohesive Property*, only reduces the size or offsets the elements using the specified value instead of the thickness value specified on the selected cohesive property.




The *Limit Region* and *Midside Node* buttons offer the same functionality they do in the *Mesh, Editing, Edge Split* command. For more information, see Section 5.3.1.4, “Mesh, Editing, Edge Split...”.

- Updated the *Mesh, Edge Member* command by dividing it into two commands which are now on the *Mesh, Edge/Skin Elements* submenu. *Mesh, Edge/Skin Elements, Line Elements on Edges* creates line elements on selected edges. *Mesh, Edge/Skin Elements, Planar Elements on Faces* creates Shell Elements on the selected faces of planar or Solid Elements.
- Updated various dialog boxes used for meshing to change the *Node Param*, *Elem Param*, *Formulation*, and *Material Orientation* buttons to icon buttons and added “tooltips” for those and for the Property icon button.
- Enhanced Mesh Points to include assigning Mesh Points on Points, Curves, and Solids, as well as Surfaces. Mesh points also now have a unique symbol that can be controlled with View Options and may be created or edited using the *Mesh Point Editor* dockable pane.
- Enhanced various meshing commands to automatically delete any extraneous planar element on surfaces after meshing surfaces which have a *Matched - Link to Surface* mesh approach assigned.
- Enhanced commands for Nodes and Elements on the *Modify, Move By* menu, the *Modify, Rotate By* menu, and the *Modify, Align* menu by adding an option to also move/rotate/align geometry associated with selected Node and Elements. In addition, added *Always Create Parent CSys* option, which can be used to force creation of a “parent” coordinate system to be used as the reference coordinate system for any coordinate systems which need to be created to maintain the proper definition of entities, such as boundary conditions or regions.
- Enhanced commands for Coordinate Systems on the *Modify, Move By* menu, the *Modify, Rotate By* menu, and the *Modify, Align* menu by adding an option to also move/rotate/align geometry and mesh associated with selected Coordinate Systems. In addition, added *Always Create Parent CSys* option, which can be used to force creation of a “parent” coordinate system to be used as the reference coordinate system for any coordinate systems which need to be created to maintain the proper definition of entities.
- Enhanced the commands on the *Mesh, Copy* menu, the commands *Mesh, Rotate* menu, and the commands on the *Mesh, Reflect* menu by adding an option to automatically select associated geometry, along with additional options.

You can use the copy/rotate/reflect commands to duplicate existing nodes or elements. These commands, other than *Mesh, Copy, Offset Element*, are similar to the *Mesh, Rotate* and *Mesh, Reflect* commands. Here, the entities are duplicated using the option specified in the *Orient/Transform* section of the *Copy Options* dialog box:




Each section of the *Copy Options* dialog box offers a number of different options, most of which can be used in conjunction with one another, to perform complex operations which can potentially provide a streamlined work-flow when copying entities. In addition, it offers tremendous flexibility in terms of what other entity types to include when creating copies of the entity type selected during the chosen command.

### *Orient/Transform - Mesh, Copy commands*

This section offers five different options to orient and/or transform copied finite element entities:

***Move Along Vector*** - Copied entities are translated along a vector specified using the standard vector definition dialog box.

***Move to Location*** - Copied entities are translated along a vector specified using standard coordinate definition dialog boxes to first specify a “Copy From” location, then a “Copy To” location. This may be useful if using different *Methods^* for selected coordinates to use as the “Copy From” and “Copy To” locations.

***Between Vectors*** - Copied entities are “aligned” using two vectors, each specified using a standard vector definition dialog box. The first vector defines the original position and orientation that will be aligned. The second vector defines the new or desired position and orientation. The entities are first translated from the origin of the first vector to the origin of the second vector, then rotated based on the angle between the vectors.

***Between Planes*** - Copied entities are “aligned” using two planes, each specified using a standard plane definition dialog box. The behavior is similar to *Between Coordinate Systems*, only each specified plane (X and Y axes) and each plane’s normal direction (Z axis) are used to determine XYZ axes to align.

***Between Coordinate Systems*** - Copied entities are “aligned” using two coordinate systems selected using the *From CSys* and *To CSys* drop-downs in the *Copy Between Coordinate Systems* dialog box. The XYZ axes of the *From CSys* coordinate system will be aligned to the XYZ axes of the *To CSys* coordinate system and copies of all the selected and associated entities will move as a rigid body to the newly aligned location. The type of coordinate system used for *From CSys* and *To CSys* does not matter, as only the axes are aligned.

***Note:*** If using a cylindrical coordinate system, the R (radial) axis is the X axis, the T (theta) axis corresponds is the Y axis, and the Z axis is the Z axis. If using a spherical coordinate system, the R (radial) axis is the X axis, the P (phi) axis is the Y axis, and the T (theta) axis is the Z axis.

### *Orient/Transform - Mesh, Rotate commands*

This section offers two different options to orient and/or transform copied finite element entities:

***Rotate Around Vector*** - Copies of entities are rotated around a vector specified using the standard vector definition dialog box, using values specified for *Rotation Angle* and *Translation Distance* specified in the *Change per Repetition* dialog box. Each copy is rotated around the axis of rotation vector by the specified angle (following right-hand rule conventions), and is translated along the axis vector by specified distance. If a non-zero value for *Translation Distance* is specified, a spiral will be created.

***Rotate to Location*** - Copies of entities are rotated around a vector specified using the standard vector definition dialog box, using two locations, a “starting location” and “ending location”, specified using standard coordinate definition dialog boxes. This option may be useful if using different *Methods^* for “starting location” and “ending location” or the *Rotation Angle* and/or *Translation Distance* are not known before starting the command.

### *Reflection Options - Mesh, Reflect commands*

This section offers only one option, *Trap Width*. A copy of any selected node or element will not be made if it is closer to the reflection plane than the specified *Trap Width*. If *Trap Width* is set to zero, all of the selected nodes will be reflected. This option is used most often when reflecting elements (to join nodes on the reflection plane), and will usually be zero when reflecting nodes.

### *Numbering*

This section offers three different options to control IDs of the duplicate entities:

***Default*** - Duplicated entities are numbered sequentially, based on original ID, and given the next available ID for each entity type.

***Block*** - Duplicated entities will have the ID of the duplicated entity with the lowest original ID, for each entity type, set to the value specified in the *To* field. Then, the constant difference between the lowest original ID and the value in the *To* field will be added to all other IDs. No sorting will take place, and no gaps will be filled. This is an easy method to maintain a numbering structure while shifting it to another level (such as making sure the copies of entities 1, 20, 50, 100, and 500 will be 1001, 1020, 1050, 1100, and 1500 by using a *To* value of 1001).

***Offset*** - Duplicated entities will be offset from the original ID of each entity using the value specified in the *By* field. No sorting will take place, and no gaps will be filled. This is an easy method to maintain a numbering structure while shifting it to another level (such as making sure the copies of entities 1, 20, 50, 100, and 500 will be 1001, 1020, 1050, 1100, and 1500 by using a *By* value of 1000).

### *Other Entities to Include*

This section contains four categories of entity types which can be included when duplicating nodes or elements. The *All On* and *All Off* buttons can be used to quickly enable or disable all the currently available categories.

***Geometry*** - When enabled, geometric entities which are associated to nodes or elements will also be duplicated. For example, when this option is enabled, if a node is associated with a curve which is part of a surface or solid, only the curve and points used to define the curve will be duplicated along with the node. If instead, a node is associated with a surface, then the surface and all of the curves and points needed to define the surface will be duplicated along with the node.

***Loads*** - When enabled, nodal and elemental loads will also be duplicated. If the *Geometry* option is also enabled, any loads on geometric entities being duplicated will also be duplicated.

***Constraints*** - When enabled, nodal constraints will also be duplicated. If all nodes used by a constraint equation are being duplicated, then the constraint equation will also be duplicated. If the *Geometry* option is also enabled, any constraints on geometric entities being copied will also be duplicated.

***Connections/Regions*** - When enabled, any full or partial regions defined by nodes or elements being copied will also be duplicated. This is the case for Connection, Fluid, NonStructural Mass, Bolt, and Rotor regions. If the *Geometry* option is also enabled, any full or partial regions defined by geometric entities being duplicated will also be duplicated. In addition, if both Connection regions of a Connector are copied, then the Connector is duplicated.

### *Property/Material Options*

This section offers four different options for defining properties and materials for duplicated entities:

***Same Properties*** - Duplicated elements simply use the same property currently referenced by the original elements. When the *Geometry* option is enabled, any property meshing attribute assigned to the original geometric entity will also be assigned to the duplicated geometric entity.

***Duplicate Properties*** - Duplicated elements will reference a duplicate of the property currently referenced by the original elements. When the *Geometry* option is enabled, any property used as a meshing attribute assigned to the original geometric entity will duplicated, then the duplicate will be assigned to the duplicated geometric entity.

***Duplicate Materials*** - Same as the *Duplicate Properties* option, only each duplicate property will reference a duplicate of the material referenced by the original property. When the *Geometry* option is enabled, same as the *Duplicate Properties* option, only each duplicate property created for use as a meshing attribute will reference a duplicate of the material referenced by the original property.

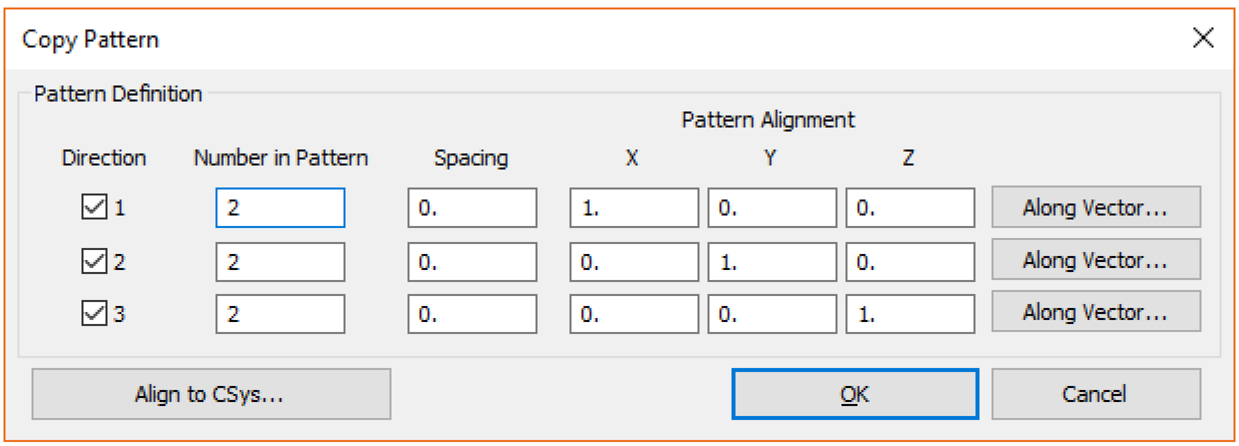
***Use Property*** - Attempts to change the property of duplicated elements to the property selected in the drop-down. Will only change the property if the duplicated element is the appropriate type to reference the selected property. When the *Geometry* option is enabled, attempts to assign the selected property as a meshing attribute to any geometric entity which had a property meshing attribute specified before being duplicated. Again, property will only be updated if it matches the property type assigned as a meshing attribute to original geometric entities.

### *Approach*

This section offers a choice between creating a *Pattern* of duplicate entities or *Single/Multiple Copies* of the selected entities. While *Pattern* may be useful in certain situations, *Single/Multiple Copies* offers some additional options, which allow for a more generalized and flexible approach when creating duplicates.

#### Pattern - Mesh, Copy commands only

When using *Pattern*, the *Copy Pattern* dialog box will be shown after clicking OK to the *Copy Options* dialog box:




The *Copy Pattern* dialog box is setup to create a copy of the originally selected entities using up to three different directions, in series. The values for *Number in Pattern* represents the “total” number of entities which will be created in each direction, while the value for *Spacing* represents the distance between each duplicate in each direction of the pattern. Duplicates of the selected entities are first created in *Direction 1*, then duplicates of the original entities and the duplicate entities created in *Direction 1* are created in *Direction 2*, then duplicates of the original entities and the duplicate entities created in *Direction 1* and *Direction 2* are created in *Direction 3*.

The *Along Vector...* buttons may be used to define a vector for any or all directions using the standard entity selection dialog box.

The *Align to CSys...* button will transform the values currently specified in the *Pattern Alignment* section to represent unit vector values in selected coordinate system.

For example, imagine copying a single selected entity with *Number in Pattern* for *Direction 1* is set to “2”, *Direction 2* is set to “3”, and *Direction 3* is set to “4”. In this case, a single copy of the selected entity is created in *Direction 1* (total entities now 2), two copies of each of those 2 entities are created in *Direction 2* (total entities now 6), then three copies of each of those 6 entities are created in *Direction 3* (total entities at end of command, 24).

#### Single/Multiple Copies

When using *Single/Multiple Copies*, a number of additional options are available.

***Repetitions*** - By default, this value is set to “1”, which creates a single duplicate of each selected entity. To create multiple duplicates of each selected entity, simply set this option to the number desired. While not required, this value should typically be set to “1” when using the *AutoRepeat Copy* option, as the command will then create a single duplicate of each selected entity at each new location.

***Copy in Same Location*** - When enabled, creates a duplicate of the selected entities without specifying any additional information for the options in the *Orient/Transform* section (i.e., creates coincident entities).

***AutoRepeat Copy*** - When enabled, repeats all dialog boxes required by the currently selected option in the *Orient/Transform* section after saying OK to the *Copy Options* dialog box (i.e., selected entities and options specified in *Copy Options* are used for all subsequent copy operations). The dialog boxes will continue to repeat until *Cancel* is pressed, thus ending the command. To avoid multiple duplicates of the selected entities being created at each new location, set *Repetitions* to “1”.

***Always Create Parent CSys*** - When enable, forces the creation of a “parent” coordinate system, which is used as the reference coordinate system for any coordinate systems which need to be created to maintain the proper definition of finite element entities, such as boundary conditions or regions.

## *Elements*

- Added *Solid Cohesive* Element Type, which are currently only used by the NX Nastran Multi-Step Nonlinear Solution Sequences (SOL 401 and SOL 402). Solid Cohesive elements can be created manually or by using the *Mesh, Editing, Cohesive Meshing* command. In addition, while they are “solid” elements, the nodes on the top and bottom faces may be coincident (i.e.,“flat”). See Meshing section.
- Added *Ansys* tab to the *Element Formulation* dialog box, which is used to specify element type (*Ansys Option* section) and up to six KEYOPT values for each element type (*Ansys Keyopt* section) for the new ANSYS translator. If the *Enable Legacy Ansys Interface* option is enabled in the *General Solver Options* section of the *Interfaces* tab of the *File, Preferences* command, the *Legacy Ansys Options* section will be available instead.

## *Materials*

- Added *NX Nastran Cohesive (MATCZ Sol 401, 402)* to *Other Types*. Used in conjunction with Cohesive Elements to model adhesion between different types of elements when using NX Nastran Multi-Step Nonlinear Solution Sequences (SOL 401 and SOL 402), as well as consider damage during progressive ply failure in models with laminate elements.

## *Properties*

- Added the *Solid Cohesive* Property Type. Used for modeling cohesion in NX Nastran Multi-Step Nonlinear Solutions (SOL 401 and SOL 402), along with damage during progressive ply failure in models with laminate elements.
- Added *Edit Layup* icon button to *Define Property - LAMINATE PLATE Element Type* and *Define Property - LAMINATE SOLID Element Type* dialog boxes, which can be used to edit the layup currently selected in the *Layup* drop-down.

## *Layups*

- Enhanced “Compute” capability to always calculate equivalent properties using multiple approaches (i.e., both including and excluding “Membrane/Bending Coupling”) and added “FOR INFORMATION PURPOSES ONLY” section, which explicitly states the calculated values “Membrane/Bending Coupling Excluded” for those values.

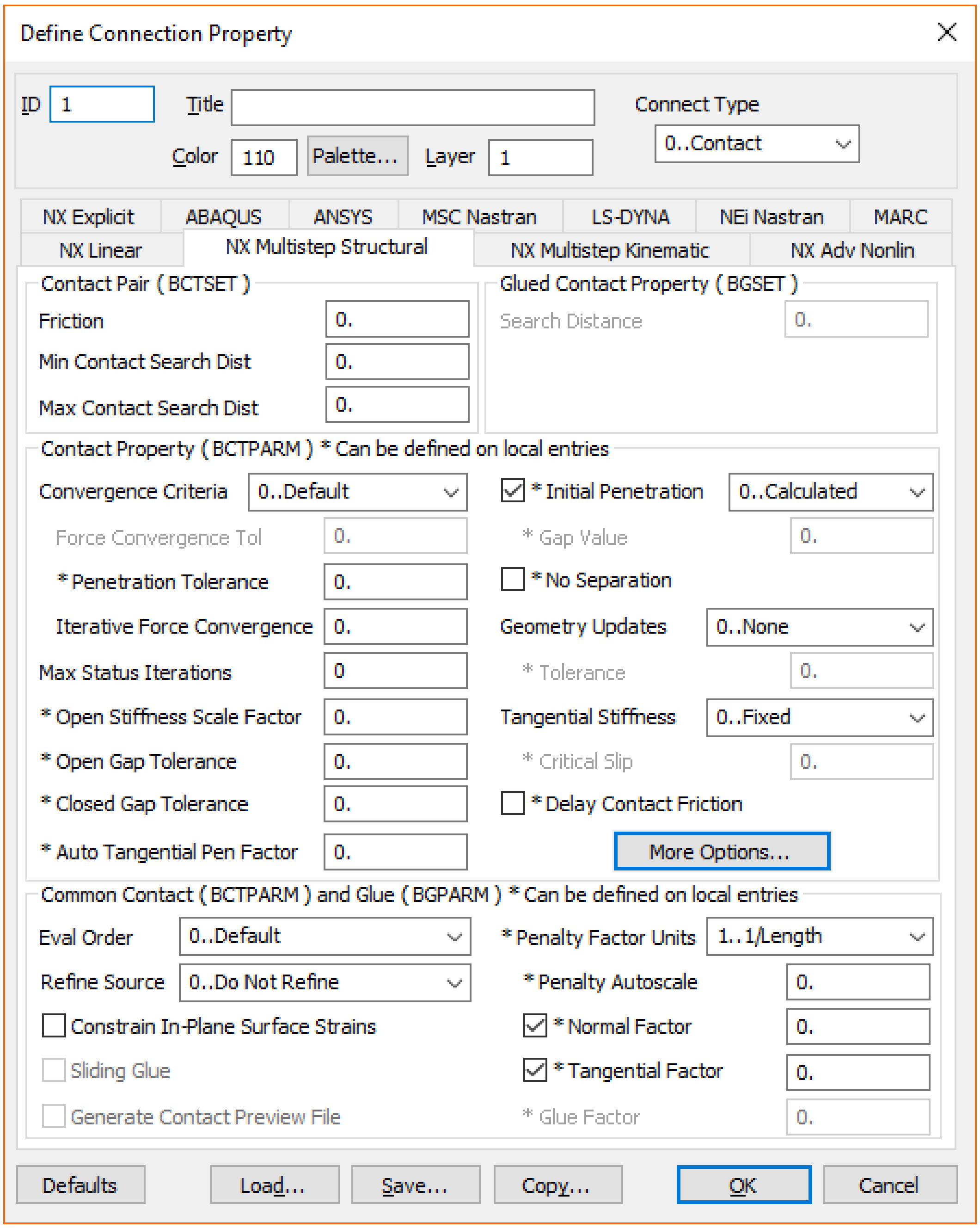
## *Loads and Constraints*

- Updated *Model, Load, Map Output From Model* command by adding *Criteria* option to *Data Conversion* section, which is designed to be used between models which have an identical or very similar mesh. When using the *Criteria* option, *Conversion Type* and *Unmapped Values* are not available.
- Updated *Model, Load, From Freebody* command, when using the *Multi-Model* option, by adding the *Re-center Reference Node* option to the *Toggle Rigid Element Nodes* dialog box. This dialog box may appear near the end of the command and is used to add or remove nodes from any of the rigid or interpolation elements automatically created by previous steps in the command.

## Connections (Regions, Properties, and Connectors)

- Added *NX Multistep Structural* tab to Define Connection Property dialog box for NX Nastran SOL 401.

For NX Nastran Solution 401, more than 1 *Connector* (contact pair) can be defined and each pair can have a different *Connection Property*. For each *Connection Region*, the values defined on the *NX Multistep Structural* tab of the *Define Connection Property* dialog box are used for each respective *Connector* (contact pair), unless "Global" or "Subcase-specific" values are defined in an Analysis Set where the *Type* is set to "27..Multi-Step Structural". A Connection Property selected in an Analysis Set can be used to set "Global" values (Contact and/or Glue), which are essentially used as the "defaults" for the entire analysis, and/or "Subcase-specific" values (Contact only), which can be used to set "defaults" for a particular subcase. These options can be reached by pressing the *NX Multistep Structural* tab in the *Define Connection Property* dialog box:



***Note:*** The *Defaults* button will automatically fill in the dialog box with the default values suggested by NX Nastran. It may be helpful to try and run the analysis with the defaults and then run it again if any modifications are needed to create more accurate results or achieve convergence.

### Contact Pair (BCTSET)

**Friction** - Enters a value in the FRICi field on the BCTSET entry. Designates the Static Coefficient of friction for contact pair "i".

**Min Contact Search Dist** - Enters a value in the MINDi field on the BCTSET entry. Designates the Minimum search distance for contact pair "i".

***Note:*** The minimum distance can be negative and used for an interference fit condition modeled as overlapping surfaces.

**Max Contact Search Dist** - Enters a value in the MAXDi field on the BCTSET entry. Designates the Maximum search distance for contact pair "i".

***Note:*** The max distance must be defined for all contact problems. This is the distance that NX Nastran will search for contact from the element normal.

### Glued Contact Property (BGSET)

**Search Distance** - Enters a value in the SDISTi field on the BGSET entry. Designates the Search Distance for the contact pair "i". Essentially, this is telling NX Nastran that if the *Connection Regions* of the *Connector* (contact pair) are within this distance, which they should be, then Glued contact will be active for this contact pair.

***Note:*** By setting the value of search distance to a value larger than the largest distance between connection regions using Glued Contact, only one Glued Contact property is needed per model. Even if you have several different properties created for Glued Contact, FEMAP will automatically combine them all in to one BGSET entry in NX Nastran.

### Contact Property (BCTPARM) *Can be defined on local entities

These options need to only be defined once for a contact analysis, regardless of how many *Connectors* (contact pairs) are defined in the model. Each *Connector* has an ID assigned to it and can reference a different *Connection Property*. FEMAP will use the *Connection Property* referenced by the *Connector* with the lowest ID to define the BCTPARM entry for the entire model.

**Convergence Criteria** - Creates the CNTCONV field on the BCTPARM entry. Specifies the Contact convergence criteria.

0..Default - Uses the default value of NX Nastran, which is to use **Penetration Tolerance* (PTOL)

1..PTOL Convergence - The contact convergence criteria is based on **Penetration Tolerance* (PTOL)

2..CTOL Convergence - The contact convergence criteria is based on *Force Convergence Tol* (CTOL)

**Force Convergence Tol** - Creates the CTOL field on the BCTPARM entry. Contact augmentation traction convergence used when *Convergence Criteria* is set to "1..PTOL Convergence". (Default = 0.05)

The augmentation loop convergence criteria can be based on traction convergence. The contact force ratio FRAT is determined as:

$$FRAT = \sqrt{\frac{((\lambda^k - \lambda^{k-1}) * (\lambda^k - \lambda^{k-1})}{(\lambda^k * \lambda^k)}}$$

where k is the augmentation loop id. If FRAT < CTOL, the contact augmentation loop is considered converged.

***Penetration Tolerance** - Creates the PTOL field on the BCTPARM entry. Contact penetration tolerance used when *Convergence Criteria* is set to “0..Default” or “1..PTOL Convergence”. (Default value = 0.01*characteristic length for each Connector).

If the contact penetrations exceed the penetration tolerance, an extra augmentation loop is performed. If the penetrations are below this tolerance, the augmentation loop is considered converged. In addition, if the global solution convergence criteria is satisfied, then the time step is considered converged.

For PTOL > 0.0: Penetration tolerance = PTOL * characteristic length for each Connector

For PTOL < 0.0: Penetration tolerance = ABS(PTOL) for each Connector

**Iterative Force Convergence** - Creates the RCTOL field on the BCTPARM entry. Iterative contact force convergence. (Default = 0.05)

**Max Status Iterations** - Creates the MAXS field on the BCTPARM entry. Maximum number of augmentation (outer) loops. If the augmentation loop has not converged in MAXS number of iterations, the solution will proceed to the next step if the global convergence criteria has been met. Setting MAXS=1 selects a pure penalty formulation. (Default = 20)

***Open Stiffness Scale Factor** - Creates the OPNSTF field on the BCTPARM entry. Open contact stiffness scale factor. (Default = 1.0E-6)

The open contact stiffness is computed by OPNSTF * closed stiffness.

***Open Gap Tolerance** - Creates the OPENTOL field on the BCTPARM entry. Open gap tolerance scale factor. (Default=1.0)

For OPENTOL > 0.0, the open contact stiffness (OPNSTF * closed stiffness) is applied to the contact elements that have a gap value less than or equal to OPNTOL * characteristic length, but greater or equal than **Closed Gap Tolerance* (GAPTOL) * characteristic length. The contact element stiffness is 0.0 if the gap is greater than OPNTOL * characteristic length.

For OPENTOL < 0.0, the open contact stiffness (OPNSTF * closed stiffness) is applied to the contact elements that have a gap value less than or equal to ABS(OPNTOL), but greater or equal than **Closed Gap Tolerance* (GAPTOL) * characteristic length. The contact element stiffness is 0.0 if the gap is greater than ABS (OPNTOL).

***Closed Gap Tolerance** - Creates the GAPTOL field on the BCTPARM entry. Closed gap tolerance scale factor. (Default = 1.0E-10)

The closed contact stiffness is applied to the contact elements that have a gap less than GAPTOL * characteristic length.

***Auto Tangential Pen Factor** - Creates the TANSCL field on the BCTPARM entry. Scales the automatically calculated tangential penalty factor, PENT, either up or down. (Default = 1.0)

This can be used to scale the stiffness of specific contact pairs if convergence issues occur. Only used when the **Tangential Factor* (PENT) option is enabled and has a value greater than 0.

***Initial Penetration** - Creates the INIPENE field on the BCTPARM entry. Use when the goal is for a pair of contact regions to be touching without interference, but due to the faceted nature of finite elements around curved geometry, some of the element edges or faces may have a slight gap or penetration. (Default = 0)

0..Calculated - Contact is evaluated exactly as the geometry is modeled. No corrections will occur for gaps or penetrations.

2..Calculated/Zero Penetrations - Penetrations will be reset to a new initial condition in which there is no interference.

3..Zero Gap/Penetration - Gaps and penetrations are both reset to a new initial condition in which there is no interference.

INIPENE is applied when contact elements are initially created, and if they are recreated as a result of large displacement effects when Large Deformations (PARAM,LGDISP,1) is defined. If the check box in from of **Initial Penetration* is not enabled, then nothing for INIPENE will be written to the BCTPARM.

***Gap Value*** - Only available when **Initial Penetration* is set to "3..Zero Gap/Penetration". Creates the GAPVAL field on the BCTPARM entry. Optionally defines a constant gap distance between the source region and the target region when INIPENE=3. A negative GAPVAL value indicates that an initial penetration exists that the software will eliminate. (Default = 0.0)

***No Separation*** - Creates the NOSEP field on the BCTPARM entry. No separation contact option.

When this box is not checked in FEMAP, no special field will be written to the BCTPARM and when contact stiffness is recomputed in a consecutive nonlinear iteration, contact elements which are inactive as a result of normal tractions = 0.0 and no penetration, and which have a gap greater than **Closed Gap Tolerance* (GAPTOL) * characteristic length will remain inactive in the consecutive iteration. This is the default behavior.

When this box is checked, The open contact stiffness (OPNSTF * closed stiffness) is applied to the inactive contact elements that have a gap value less than or equal to **Open Gap Tolerance* (OPNTOL) * characteristic length, but greater or equal than **Closed Gap Tolerance* (GAPTOL) * characteristic length. The contact elements with a gap greater than OPNTOL * characteristic length remain inactive. While sliding is permitted with this option, the magnitude of the sliding can be controlled by the tangential penalty factor. To define frictionless sliding, set the coefficient of friction=0.0 or **Tangential Factor* (PENT) = 0.0.

**Geometry Updates** - Creates the GUPDATE field on the BCTPARM entry. Geometry update flag.

-1..Default - When PARAM, LGDISP=1 uses "2..Start, SLIP>(GUPTOL*Avg Element Length)". When PARAM, LGDISP=-1, uses "3..Once Per Step".

0..None - Contact geometry updates will not be done during the analysis.

1..SLIP>(GUPTOL*Avg Element Length) - Contact geometry updates occur when SLIP > **Closed Gap Tolerance* (GUPTOL) *Average element length.

2..Start, SLIP>(GUPTOL*Avg Element Length) - Contact geometry updates occur at the start of each step and when SLIP > **Closed Gap Tolerance* (GUPTOL) *Average element length.

3..Once per Step - Contact geometry updates occur once a step.

4..Each Iteration - Contact geometry updates occur every iteration.

***Tolerance*** - Only available when *Geometry Updates* is set to "1..SLIP>(GUPTOL*Avg Element Length)" or "2..Start, SLIP>(GUPTOL*Avg Element Length)". Creates the GUPTOL field on the BCTPARM entry. Geometry update tolerance. (Default = 0.25*characteristic length)

If the relative sliding distance between the source and target regions exceeds this tolerance, a geometry update will be initiated with large displacement.

**Tangential Stiffness** - Creates the FRICMOD field on the BCTPARM entry. Tangential stiffness selection. (Default =1..Adaptive)

0..Fixed - The tangential stiffness is fixed throughout the subcase.

1..Adaptive - The tangential stiffness is adaptively modified at every iteration as a function of contact pressure.

When using "1..Adaptive", the tangential stiffness is computed as:

(FRICi * contact pressure) / SCRIT

...where FRICi is the coefficient of friction defined on the BCTSET entry (*Friction*), and SCRIT is the value specified for **Critical Slip*.

***Critical Slip*** - Only available when *Tangential Stiffness* is set to "1..Adaptive". Creates the SCRIT field on the BCTPARM entry. Defines the critical slip value. (Default=0.005)

For SCRIT > 0.0: Critical slip = SCRIT * characteristic length for each contact pair.

For SCRIT < 0.0: Critical slip = ABS(SCRIT) for each pair.

***Delay Contact Friction*** - Creates the FRICDLY field on the BCTPARM entry. Option to delay contact friction to help alleviate convergence problems.

When this box is not checked in FEMAP, no special field will be written to the BCTPARM and friction is not delayed. This is the default behavior.

When this box is checked, friction is delayed and is not included in the solution until the second contact iteration.

### More Options... button

Clicking the *More Options...* button will bring up the *NX Multistep Nonlinear Options* dialog box, which allows specification of additional options for contact in SOL 401.

**Disp Scaling** - Creates the DISCAL field on the BCTPARM entry. Displacement scaling option.

When this option is enabled in FEMAP, no special field will be written to the BCTPARM and scaling will be done if required during every iteration. A check will be performed after every displacement increment to see if the incremental displacement would cause penetration between the source and target regions. If the penetrations exceed *Tolerance* in the *NX Multistep Nonlinear Options* dialog box (DISTOL), the entire incremental displacement will be scaled back to limit the penetrations in the model. This is the default behavior.

When this option is disabled, no scaling will be done. Writes DISCAL with a value of 0 to the BCTPARM.

**Tolerance** - Creates the DISTOL field on the BCTPARM entry. Tolerance for displacement scaling feature. (Default = 1.0* characteristic length)

**Stabilization Damping Option** - Creates the CTDAMP field on the BCTPARM entry. The CTDAMP parameter is available to request the stabilization damping option when you are relying on the contact condition to prevent rigid body conditions, but the contact condition is not active.

0..No Stabilization Damping - Stabilization Damping is not considered. This is default behavior.

1..Stabilization Damping in First Subcase - Stabilization damping applied only in the first subcase and it is ramped down to zero by the end of the subcase. The stabilization damping is only applied this way when the entire pair status is open.

2..Stabilization Damping when Open - Stabilization damping applied always as long as the pair status is open.

3..Stabilization Damping always On - Stabilization damping always applied, regardless of the time and the pair status.

**Normal Scale Factor** - Creates the CTDAMPN field on the BCTPARM entry. If you request stabilization damping with the *Stabilization Damping Option* (CTDAMP), the CTDAMPN parameter is available to either scale the normal damping value which the software automatically computes, or to define the normal damping value explicitly. (Default=1.0; units = pressure/velocity)

For CTDAMPN > 0.0, the normal damping value = CTDMAPN *automatically computed damping value.

For CTDAMPN < 0.0, the normal damping value = ABS(CTDMAPN).

**Tangential Scale Factor** - Creates the CTDAMPT field on the BCTPARM entry. If you request stabilization damping with the *Stabilization Damping Option* (CTDAMP), the CTDAMPN parameter is available to either scale the tangential damping value which the software automatically computes, or to define the tangential damping value explicitly. (Default=1.0; units = pressure/velocity)

For CTDAMPT > 0.0, the tangential damping value = CTDMAPT *automatically computed damping value.

For CTDAMPT < 0.0, the tangential damping value =ABS(CTDMAPT).

**Interference** - Creates the INTRFC field on the BCTPARM entry. Selects the interface behavior between the source and target region.

0..No Stabilization Damping - No interaction between source and target region.

1..Standard - Standard segment-segment contact algorithm with large sliding (pairing updates) and friction. This is the default behavior

2..No Sliding - Does not allow sliding by assuming infinite friction. Source and target regions can separate.

3..Bonded/Tied - No relative displacement is permitted in both normal and tangential directions.

4..No Separation - No relative displacement in normal direction. Frictionless small sliding (no pairing updates) in tangential directions

5..Small Sliding - Simplified contact algorithm with no pairing updates or friction.

**Ramp Initial Penetrations** - Creates IPENRAMP field in BCTPARM entry. Requests that the software ramp the removal of initial penetrations when **Initial Penetration* is set to “0..Calculated”. Initial penetrations can exist due to geometry or mesh irregularities between source and target faces or edges.

When this box is not checked in FEMAP, no special field will be written to the BCTPARM and no ramping is applied in the constant time subcase. This is the default behavior.

When this box is checked, the initial penetrations are ramped by the load factor.

**Include Shell Thickness Offset** - Creates SHLTHK field in BCTPARM entry. Shell thickness offset flag.

When this box is checked in FEMAP, no special field will be written to the BCTPARM and half shell thickness is used as surface offset. This is the default behavior.

When this box is not checked, shell thickness is excluded and contact is evaluated at mid-plane.

**Include z-Offset in Contact** - Creates ZOFFSET field in BCTPARM entry. Determines if the shell element z-offset is included

When this box is not checked in FEMAP, no special field will be written to the BCTPARM and the shell z-offset is included when determining the contact surfaces. This is the default behavior.

When this box is not checked, the shell z-offset is not included when determining the contact surfaces.

**Stiffness Stabilization** - Creates KSTAB field in BCTPARM entry. Stiffness stabilization for contact.

When this box is not checked in FEMAP, no special field will be written to the BCTPARM and stiffness stabilization is off. This is the default behavior.

When this box is checked, The stiffness matrix is stabilized when it is singular due to inactive contact constraints. The stabilization adds a factor (1.0) to the diagonal terms of the stiffness matrix. KSTAB=1 is only supported with the sparse solver, and will disable any open contact stiffness specified through the **Open Stiffness Scale Factor* (OPNSTF).

### Common Contact (BCTPARM) and Glue (BGPARM) *Can be defined on local entities

This section contains options for Glued contact and several options available for both Glued Contact and Linear Contact.

**Eval Order** - Creates INTORD field in BCTPARM or BGPARM entry. Determines the number of “Linear Contact or Glue Points” for a single element on the source region.

0..Default - Does NOT write the INTORD field or corresponding value field to the BCTPARM or BGPARM entry. Simply uses the default value for “Linear” or “Glued” contact built into the NX Nastran solver.

1..Low - Lowest order of points on source region.

2..Medium - Medium order of points on source region. This is the default behavior.

3..High - Highest order of points on source region.

The higher the integration order, the longer the solve will take.

**Refine Source** - Creates REFINE field on the BCTPARM or BGPARM entry. Determines if the source region is refined for the “Linear” or “Glued” Contact solution.

0..Do Not Refine - Does not refine the “Linear Contact/Glue” source region based on target surface definition.

2..Refinement Occurs - Refines the “Linear Contact/Glue” source region based on target surface definition. This is default behavior.

**Constrain In-Plane Surface Strains** - Only available when *Connect Type* is set to “1..Glued”. Creates ESOPT field on the BGPARM entry. Can be used to change the edge-to-surface glue stiffness. Shell theory does not account for changes in shell thickness or normal strains perpendicular to the plane of the shell element. This parameter gives you a choice of how to handle the linking of the zero normal strains which exist in the shell element associated with the edge, to the surface being glued.

When this box is not checked in FEMAP, no special field will be written to the BGPARM and strains in the plane of the surface being glued in the direction perpendicular to the edge are not constrained by the glue stiffness. This is the default for NX Nastran.

When this box is checked, the strains in the plane of the surface being glued in the direction perpendicular to the edge are constrained by the glue stiffness.

**Sliding Glue** - Only available when *Connect Type* is set to "1..Glued" Creates SLIDE field on the BGPARM entry. This is a flag to request the export of a bulk data representation of the element edges and faces where glue elements are created.

When this box is not checked, no special field will be written to the BGPARM. This is the default behavior.

When this box is checked, the bulk data export occurs.

**Generate Contact Preview File** - Creates PREVIEW field on the BCTPARM or BGPARM entry. This is a flag to request the export of a bulk data representation of the element edges and faces where glue elements are created.

When this box is not checked, no special field will be written to the BCTPARM or BGPARM and the bulk data export does not occur. This is the default behavior.

When this box is checked, the bulk data export occurs.

**Penalty Factor Units** - Creates the PENTYP field on the BCTPARM or BGPARM entry. Specifies how contact element stiffness is calculated for GLUETYPE=2, which is used for all glued contact in SOL 401.

1..Scale Factor - Glue Factor (PENGLUE) is a unitless value.

2..F/L^2 - Glue Factor (PENGLUE) has the units of F/Length squared.

***Penalty Autoscale -** Creates the AUTOSCAL field on the BCTPARM entry. Scales the automatically calculated penalty normal penalty factor, PENN, either up or down. (Default = 1.0)

This can be used to scale the stiffness of specific contact pairs if convergence issues occur. Only used when the **Normal Factor* (PENN) option is enabled and has a value greater than 0.

***Normal Factor** - Creates the PENN field on the BCTPARM entry. Designates the penalty factor for the normal direction.

When this box is not checked, no special field will be written to the BCTPARM and the Normal Penalty Factor is automatically calculated. This is the default behavior.

When this box is checked and specified value is greater than 0.0, value is used as the Normal Penalty Factor.

***Note:*** If **Tangential Factor* option is enabled and has a value greater than 0.0, which defines PENT, but **Normal Factor* option is not enabled, then PENN is automatically calculated (PENN = 10*PENT)

***Tangential Factor** - Creates the PENT field on the BCTPARM entry. Designates the penalty factor for the tangential direction.

When this box is not checked, no special field will be written to the BCTPARM and the Tangential Penalty Factor is automatically calculated. This is the default behavior.

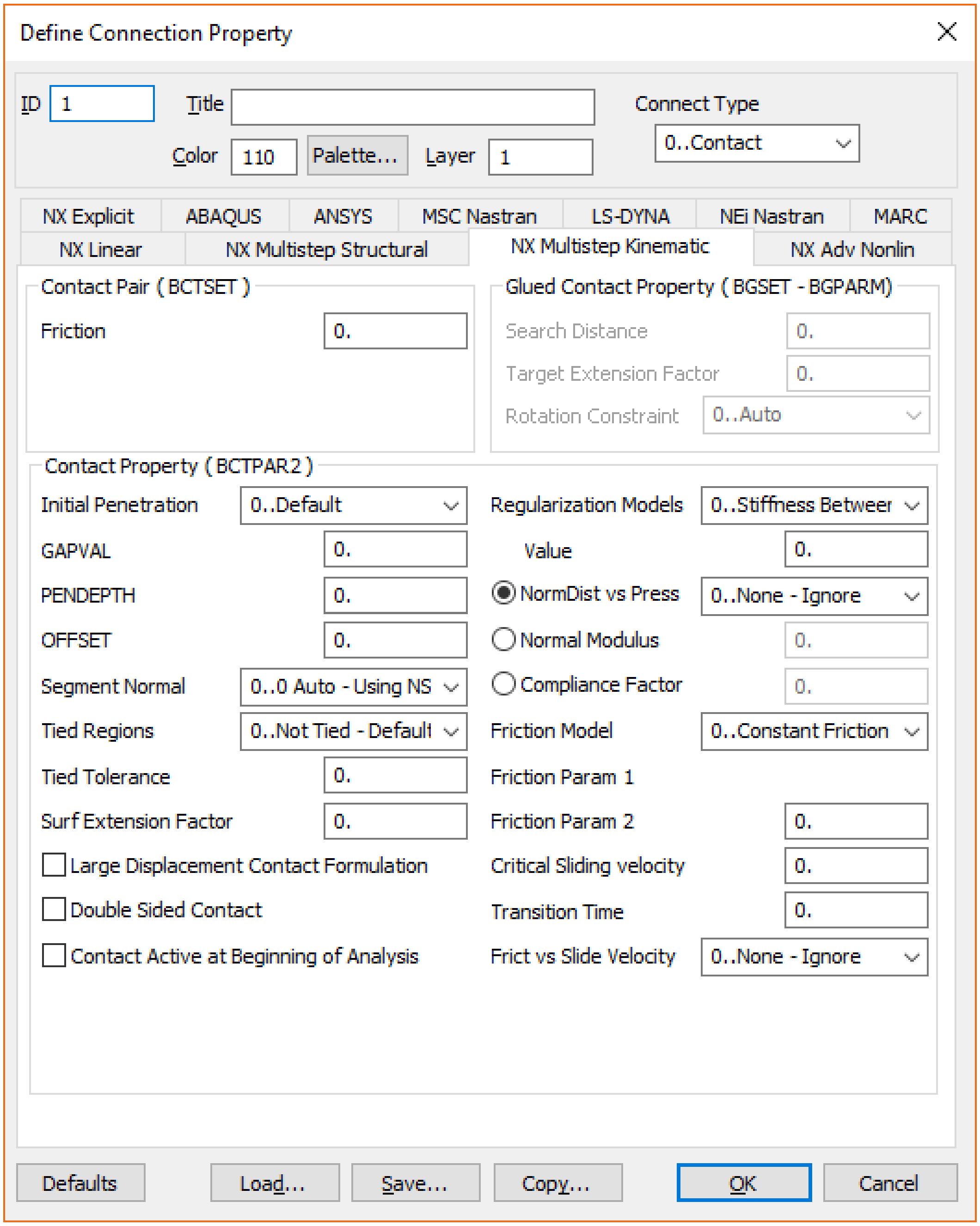
When this box is checked and specified value is greater than 0.0, value is used as the Tangential Penalty Factor.

***Note:*** If **Normal Factor* option is enabled and has a value greater than 0.0, which defines PENN, but **Tangential Factor* option is not enabled, then PENT is automatically calculated (PENT = PENN/10)

***Glue Factor** - Only available when *Connect Type* is set to "1..Glued". Creates the PENGLUE field on the BGPARM entry. Designates penalty factor for GLUETYPE=2, which is used for all glued contact in SOL 401. (Default = 1.0)

- Added *NX Multistep Kinematic* tab to *Define Connection Property* dialog box for NX Nastran SOL 402.

For NX Nastran Solution 402, more than 1 *Connector* (contact pair) can be defined and each pair can have a different *Connection Property*. For each *Connection Region*, the values defined on the *NX Multistep Kinematic* tab of the *Define Connection Property* dialog box are used for each respective *Connector* (contact pair), unless “Global” or “Subcase-specific” values are defined in an Analysis Set where the *Type* is set to “28..Multi-Step Nonlinear Kinematic”. A Connection Property selected in an Analysis Set can be used to set “Global” values (Contact and/or Glue), which are essentially used as the “defaults” for the entire analysis. These options can be reached by pressing the *NX Multistep Kinematic* tab in the *Define Connection Property* dialog box:




### Contact Pair (BCTSET)

**Friction** - Enters a value in the FRICi field on the BCTSET entry. Designates the Static Coefficient of friction for contact pair “i”.

### Glued Contact Property (BGSET - BGPARM)

These options are only available when Connect Type is set to “1..Glued”.

**Search Distance** - Enters a value in the SDISTi field on the BGSET entry. Designates the Search Distance for the contact pair “i”. Essentially, this is telling NX Nastran that if the *Connection Regions* of the *Connector* (contact pair) are within this distance, which they should be, then Glued contact will be active for this contact pair.

***Note:*** By setting the value of search distance to a value larger than the largest distance between connection regions using Glued Contact, only one Glued Contact property is needed per model. Even if you have several different properties created for Glued Contact, FEMAP will automatically combine them all in to one BGSET entry in NX Nastran.

**Target Extension Factor** - Enters a value in the EXTi field specified on the BGSET entry for the contact pair “i”. Specifies an “extension factor” for the target region. The EXTi extension value is equal to the element size in the parameter space -1 to 1, or 0 to 1, depending on the element type (0.0 < Real < 0.25). For example, a value of 0.01 corresponds to 1% of the element size in all directions. (Default = 0.01)

**Rotation Constraint** - Creates BGOPT field in BGPARM entry. Source node rotation constraint option, SOL 402.

0..Auto - The best option is automatically chosen according to the source and target regions defined for the glue pair. This is the default behavior.

1..Source Rotations Unconstrained - For the nodes in the source region, the rotations are not constrained.

2..Source Rotations Constrained with Target Rotations - For the nodes in the source region, the rotations are constrained using target nodal rotations. For instance, this option can be used for when using glue between two regions of shell elements.

3..Source Rotations Constrained with Target Displacements - For the nodes in the source region, the rotations are constrained using target mean nodal displacements. This option can be used for instance when you glue shells to faces of volumes.

### Contact Property (BCTPAR2)

**Initial Penetration** - Creates the INIPENE field and corresponding value on BCTPAR2 entry. Use when the goal is for a pair of contact regions to be touching without interference, but due to the faceted nature of finite elements around curved geometry, some of the element edges or faces may have a slight gap or penetration. (Default = 1)

0..Default - Will not write the INIPENE field and value to BCTPAR2, thus used the default in NX Nastran

1..Initial Penetrations - Eliminated - Initial penetrations are eliminated and the list of penetrating nodes is printed. (Minimum and maximum penetrations are always written for each contact pair at the beginning of the computation).

2..Initial Penetrations - Ignored - Initial penetrations are ignored. In successive steps, each contactor node is allowed to penetrate the target up to its initial penetration.

3..Initial Penetrations - GAPVAL Override -Initial penetrations or gaps are overridden by GAPVAL.

**GAPVAL** - Writes the GAPVAL field and specified value to the BCTPAR2 entry. Specifies a constant gap distance between the source region (contactor) and the target region when INIPENE = 3. Negative GAPVAL means that initial penetrations will be eliminated (Default = 0.0; Unit = Length).

**PENDEPTH** - Writes the PENDEPTH field and specified value to the BCTPAR2 entry. Distance (penetration or gap) used to limit contact detection for target contact surface. (Default=5 * the size of the greatest target face; Unit = Length). If PDEPTH > 0.0, then contact element is created only if the distance (penetration or gap) is less than or equal to PDEPTH.

**OFFSET** - Writes the OFFSET field and specified value to the BCTPAR2 entry. Default offset distance for contact regions. (Default=0.0; Unit = Length)

**Segment Normal** - Creates the SEGNORM field and corresponding value on the BCTPAR2 entry. Indicates whether a continuous (interpolated) contact segment normal (and associated smoothed surface) is used for the contact surfaces. (Default=0)

-1..Disable Continuous Segment Normal - Continuous segment normal is not used.

0..Auto - Using NSIDE - SEGNORM=1 if NSIDE=1, SEGNORM=-1 if NSIDE=2.

1..Continuous Segment Normal - Continuous segment normal is used.

**Tied Regions** - Creates the TIED field and corresponding value on BCTPAR2 entry. Indicates whether contact regions in each contact pair are tied together in the normal direction. (Default=0)

0..Not Tied - Default - Contact regions are not tied.

1..Tied - Contact Regions are tied.

2..Tied After Contact - Contact regions remain tied after contact is active.

**Tied Tolerance** - Writes the TIEDTOL field and specified value to the BCTPAR2 entry. Tolerance used to determine whether contact nodes are tied to the target region when TIED=1 is specified. A contactor node is tied to its target region if the distance between them is less than or equal to TIEDTOL. (Default=0.0; Unit = Length)

**Surf Extension Factor** - Writes the EXTFAC field and specified value to the BCTPAR2 entry. Factor for extending contact surfaces beyond their boundaries. The amount of extension is given by this factor multiplied by the length of the contact segments (Must be greater than or equal to 0.0; No default; Unitless).

**Large Displacement Contact Formulation** - When enabled, writes the DISP field with a value of “2” to the BCTPAR2 entry.

**Double Sided Contact** - When enabled, writes the NSIDE field with a value of “2” to the BCTPAR2 entry, which means contact surfaces are double-sided. When disabled, nothing is written to the BCTPAR2, which means contact surfaces are single-sided, which is the default.

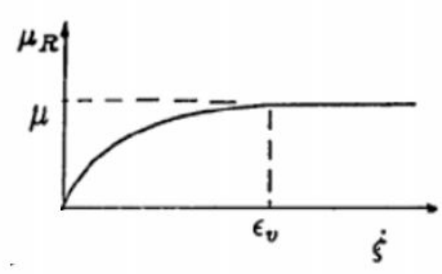
**Contact Active at Beginning of Analysis** - When enabled, writes nothing out to the BCTPAR2 entry, which means contact is active from the beginning of the simulation. This is the default behavior. If a contact pair is taken into account in the first subcase and if time assigned loads are ramped (either by default or because the parameter NLCNTL2,LVAR is set to RAMP), indicates if contact pairs are active from the beginning of the simulation. When disabled, writes ACTIVE with a value of “0”, which means contact is not considered when the simulation starts.

**Regularization Models and Value** - Depending on the option selected in the Regularization Models drop-down, creates either the STFR or the TOL field and corresponding value on BCTPAR2 entry. To avoid convergence problems when the contact starts sliding, it is recommended to introduce a regularization, either in terms of displacements or sliding velocity. This ensures a smooth transition of the friction stress between sliding and no sliding.

0..Stiffness Between the Friction Stress and the Relative Displacement - Stiffness between the friction stress and the relative displacement. When the friction stress becomes equal to the pressure multiplied by the friction coefficient, this stiffness has no more influence. (Default=1.0E8; Unit =Pressure/Length).

1..Regularized Friction Coefficient vs Sliding Velocity - Use a regularized friction coefficient μR as a function of the sliding velocity $\dot{\xi}$

$$\mu_R(\dot{\xi}) = \begin{cases} \mu\left(2-\dfrac{|\dot{\xi}|}{\varepsilon_v}\right)\dfrac{\dot{\xi}}{\varepsilon_v} & |\dot{\xi}| < \varepsilon_v \\ \mu\dfrac{\dot{\xi}}{|\dot{\xi}|} & |\dot{\xi}| \geq \varepsilon_v \end{cases}$$




where ε v = TOL (no default; Unit =Length/Time).

Once the sliding velocity becomes greater than or equal to the critical sliding velocity TOL, this regularization has no more influence. Depending on the friction model, this friction coefficient can have additional dependencies with the sliding velocity, time, and/or temperature.

***Note:*** The “1..Regularized Friction Coefficient vs Sliding Velocity” option is available for ANALYSIS=DYNAMICS subcases only. It is not available for static analyses because the velocities must be computed by the solver, or when the friction coefficient is infinite (FRICMOD=2).

**NormDist vs Press** - When this option is selected, choose an existing function from the drop-down, which describes the relation between the normal distance and the contact pressure. This will write the NPENAL, along with the ID of the selected function, to the BCTPAR2. For the function, the units for the X-Axis are Length and units for the Y-Axis, Pressure.

**Normal Modulus** - When this option is selected, enter a value for normal contact modulus, which will be written as NCMOD to the BCTPAR2 entry. (Default = Infinite, Units = Pressure/Length)

**Compliance Factor** - When this option is selected, enter a value for compliance factor, which will be written as CFACTOR1 to the BCTPAR2 entry. (Default = 0.0, Units = Length$^3$/Force)

***Note:*** *NormDist vs Press*, *Normal Modulus*, and *Compliance Force* are mutually exclusive, thus, only the option which is currently selected will be exported to NX Nastran.

**Friction Model** - Creates the FRICMOD field and corresponding value on BCTPAR2 entry. Type of friction model (Default = 0). Used in conjunction with *Friction Param 1*, *Friction Param 2*, *Critical Sliding Velocity*, *Transition Time*, and/or *Frict vs Slide Velocity* function which may also be written to the BCTPAR2 entry.

0..Constant Friction - Constant Friction, which uses the friction coefficient FRICi specified for each contact pair in the BCTSET entry. All friction parameters on BCTPAR2 are ignored.

1..Time, Velocity or Temperature Dependent - Friction depends on time, velocity, or temperature. Only *Friction Param 1* and *Frict vs Slide Velocity* are used.

2..Infinite Friction - Uses an infinite friction coefficient. All friction parameters on BCTPAR2 and the friction coefficient FRICi specified for each contact pair in the BCTSET entry are ignored.

3..Sliding Velocity Dependent - Friction depends on sliding velocity. Only *Friction Param1*, *Friction Param 2*, and *Critical Sliding Velocity* are used.

4..Linear Law of Sliding Velocity Dependent - Friction depends on a linear law of the sliding velocity. Only *Friction Param1*, *Friction Param 2*, and *Critical Sliding Velocity* are used.

5..Linear Law of Time Dependent - Friction depends on the linear law of the time. Only *Friction Param1*, *Friction Param 2*, and *Transition Time* are used.

***Note:*** Please consult the BCTPAR2 entry in the Quick Reference Guide for NX Nastran 12 or above for an explanation of the various friction models.

**Friction Param 1** - Writes the FPARA1 field and specified value to the BCTPAR2 entry. Used by Friction Models 1, 3, 4, and 5 (Unitless)

**Friction Param 2** - Writes the FPARA2 field and specified value to the BCTPAR2 entry. Used by Friction Models 3, 4, and 5 (Unitless)

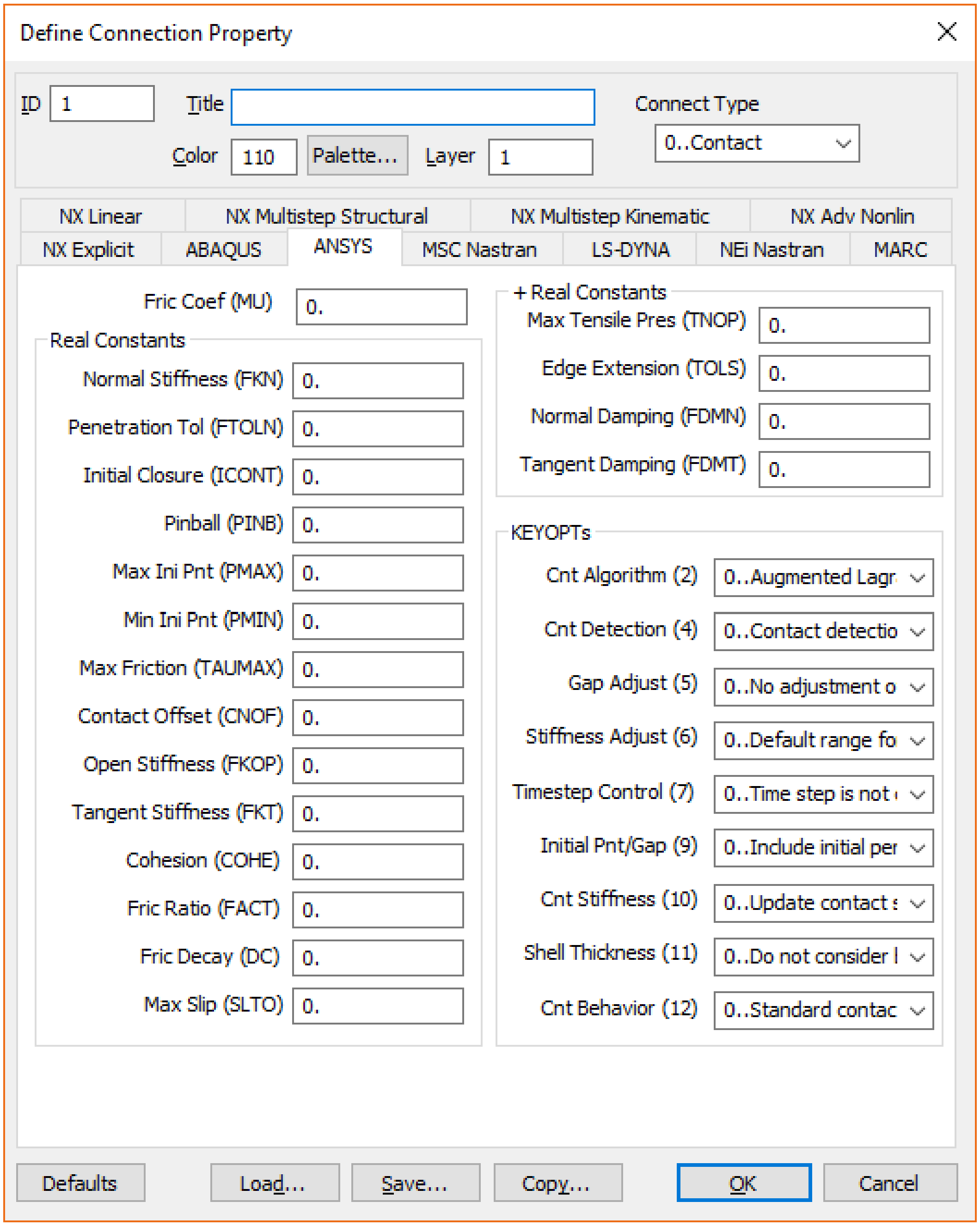
**Critical Sliding Velocity** - Writes the VCRIT field and specified value to the BCTPAR2 entry. Used by Friction Models 3 and 4 (Units = Length/Time)

**Transition Time** - Writes the TCRIT field and specified value to the BCTPAR2 entry. Used by Friction Model 5 (Units = Time)

**Frict vs Slide Velocity** - Choose an existing function from the drop-down, which describes the relation between the friction coefficient vs sliding velocity. This will write the CFVE, along with the ID of the selected function, to the BCTPAR2. For the function, the units for the X-Axis are Length/Time and units for the Y-Axis, Unitless. Used by Friction Model 1.

- Added new *ANSYS* tab to *Define Connection Property* dialog box.

The ANSYS-specific tab allows you to specify contact options for ANSYS:




The *Fric Coef (MU)* field is used to specify friction between the contact pair.

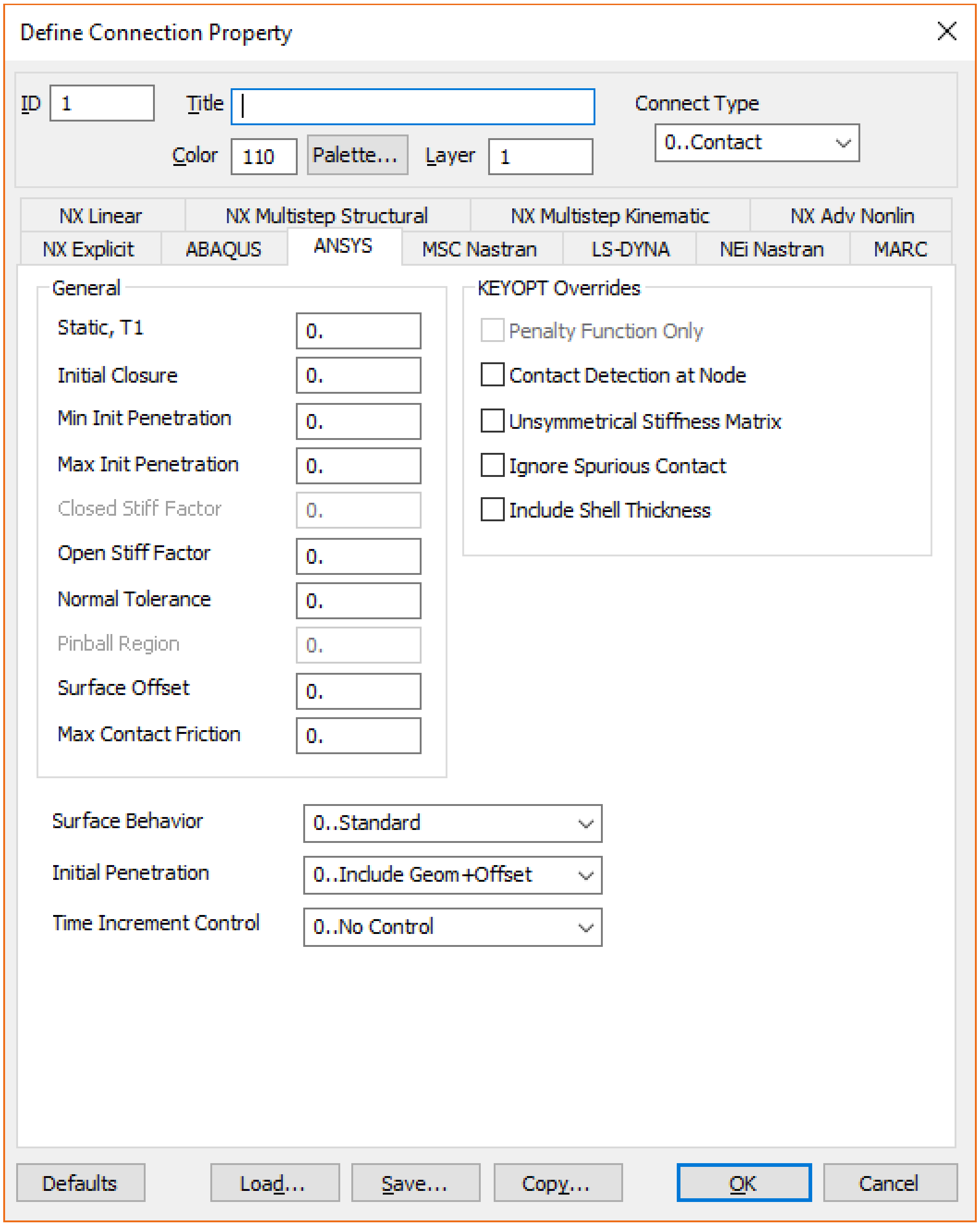
The items in the *Real Constants* and *+ Real Constants* sections allow specification of the real constants to be used by the TARGE169 (2-D), TARGE170 (3-D), CONTA171 (2-D), CONTA172 (2-D with midside nodes), CONTA173 (3-D), and CONTA174 (3-D with midside nodes). Contact surface elements are associated with target segment elements through a shared set of real constants, and ANSYS only looks for contact between surfaces with the same real constant set. Only contact elements and target elements of the same dimension (2-D or 3-D) can be in contact with each other.

This items in the KEYOPTs section, can be used to specify additional contact parameters. All of these parameters correspond to KEYOPT entries on the ANSYS contact and target elements. These are more advanced options used to create contact models which require additional parameters and the number in parenthesis after the text corresponds to the ID of the KEYOPT being specified with the accompanying drop-down.

Be sure to review the ANSYS Element Reference Guide as well as the ANSYS Structural Analysis Guide before beginning any type of nonlinear contact analysis.

***Note:*** If *Connect Type* is set to “1..Glued” a good idea is to click the *Defaults* button at the bottom of the *Define Connection Property* dialog box, as this set appropriate values and settings in this dialog box.

If the *Enable Legacy Ansys Interface* option is enabled on the *Interfaces* tab of the dialog box accessed by the *File, Preferences* command, then the following dialog box will be displayed:



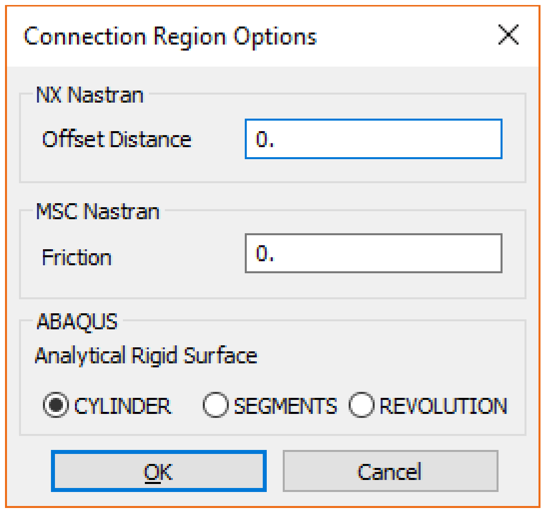

For complete definitions of these real constants and before beginning any type of nonlinear contact analysis, please review the ANSYS Element Reference Guide as well as the ANSYS Structural Analysis Guide.

This dialog can be used to specify additional contact parameters. All of these parameters correspond to KEYOPT entries on the ANSYS contact and target elements. These are more advanced options used to create contact models which require additional parameters.

The check boxes in the *KEYOPT Overrides* section of the dialog box allow you to toggle between two options for KEYOPTs (2), (4), (5), (8), and (11). The pull-down boxes in the lower portion of the dialog box correspond to KEYOPTs (7), (9), and (12), which offer additional options used to create a more realistic contact model.

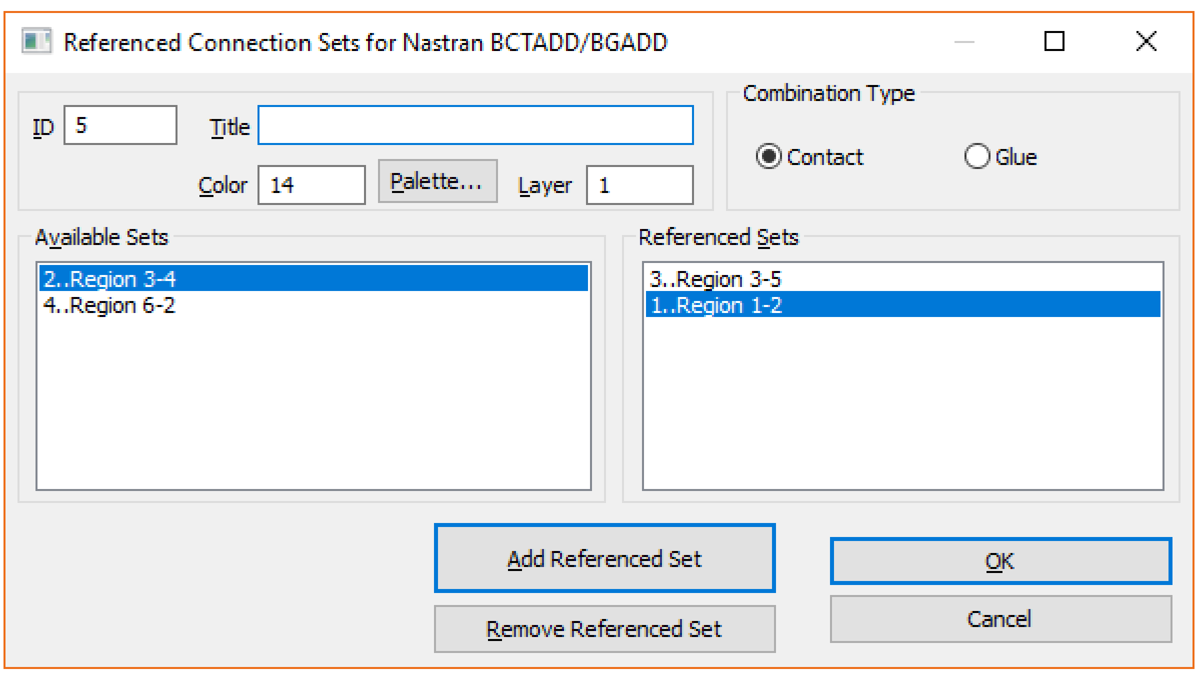
***Note:*** If *Connect Type* is set to “1..Glued” a good idea is to click the *Defaults* button at the bottom of the *Define Connection Property* dialog box. This will choose an appropriate setting for *Surface Behavior* that will create “Bonded” contact in ANSYS.

- Added an icon button to various “Define Region” dialog boxes to limit elements used for face selection. Previously, the only option was “Add Multiple” which automatically selected all visible elements and while it was possible to limit the elements for face selection from the face selection dialog box, this streamlines the process.
- Added *ABAQUS* section to the *Connection Region Options* dialog box, which is accessed via the *Region Options* button in the *Connection Region* dialog box. This option is used to specify the type of Analytical Rigid Surface, CYLINDRICAL, SEGMENTS, or REVOLUTION, to write to the ABAQUS input file.




- Added *Connect, Connector Set* command to create sets of Connectors (Contact or Glued).

Allows creation of a “set” of Connectors (contact pairs), which will write a BCTADD when *Combination Type* is set to *Contact* or a BGADD when *Combination Type* is set to *Glue*.

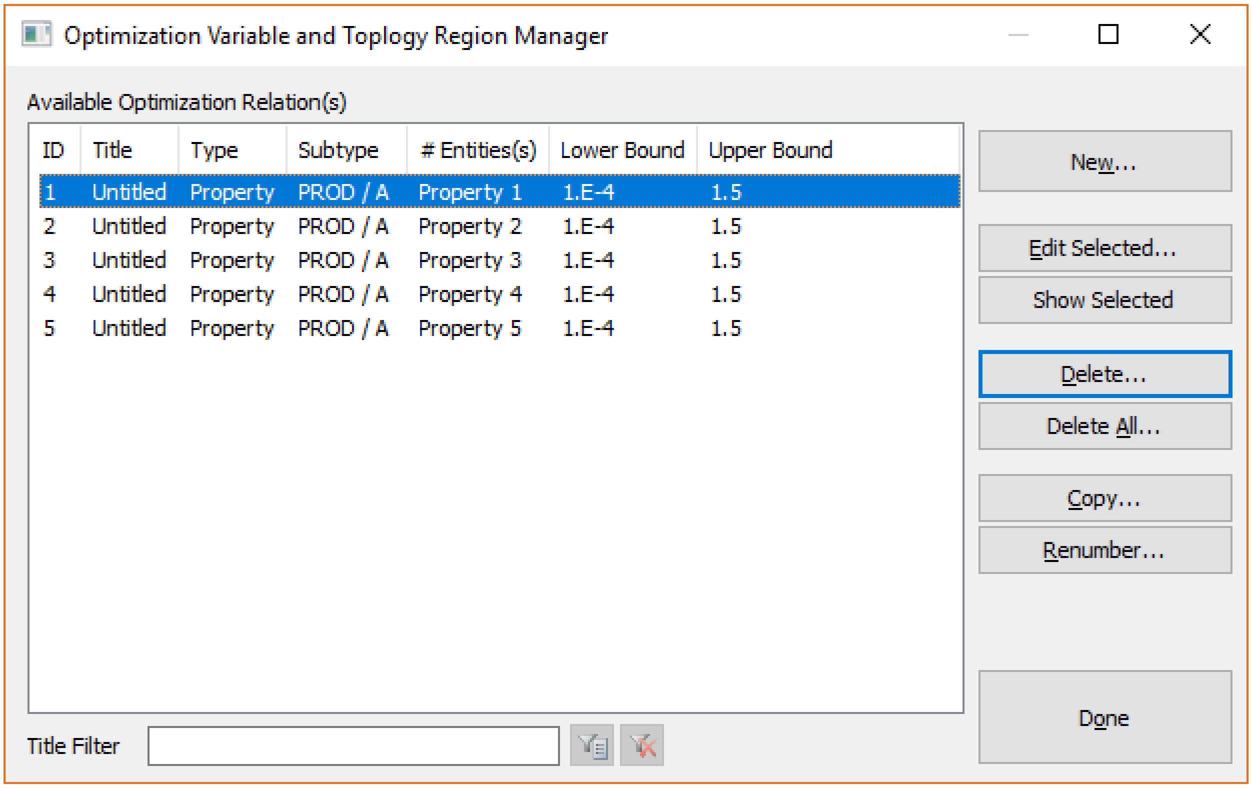


Any number of connectors may be moved between the *Available Sets* list and *Referenced Sets* list for a particular connector set by highlighting the connectors in the appropriate list and pressing the *Add Referenced Set* or *Remove Referenced Set* buttons. When *Combination Type* is set to *Contact*, only connectors which reference a connection property which has *Connect Type* set to “0..Contact” will be available to move between the two lists. When *Combination Type* is set to *Glue*, only connectors which reference a connection property which has *Connect Type* set to “1..Glue” are available. Connector sets are only used by NX Nastran Solutions Sequences SOL 401 and SOL402.

## Optimization

- Added *Model, Optimization, Variables and Topology Regions* command which opens the *Optimization Variable and Topology Region Manager*, which is used to create, edit, copy, renumber, delete selected, or delete all Optimization Variables (relationships) and/or Topology Regions.

Opens the *Optimization Variable and Topology Region Manager*:



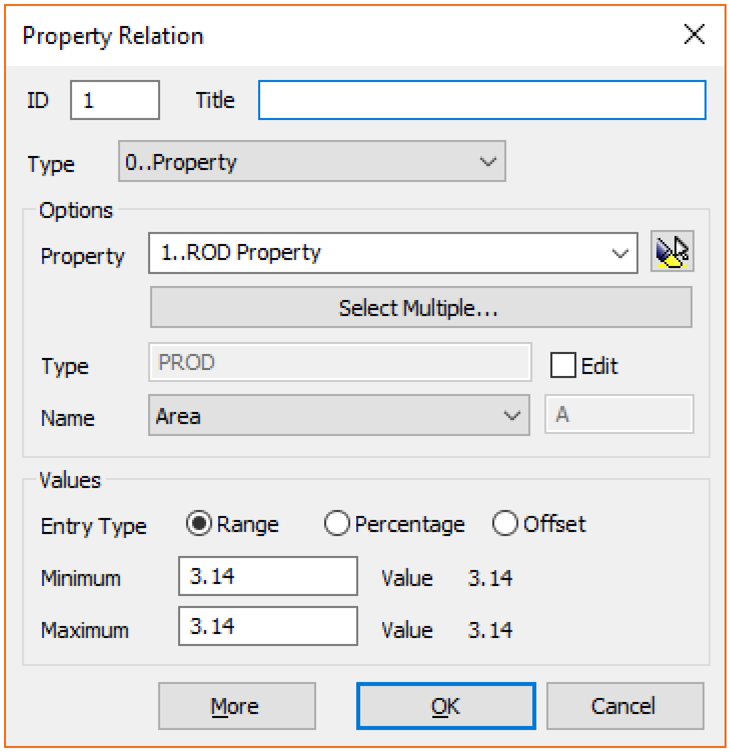

This manager is used to create design variables, or “relations”, for Properties (Type = “0..Property”), Materials (Type = “1..Material”), and/or Elements (Type = “3..Element”) for design optimization (NX Nastran or MSC Nastran) or “Topology Regions” (Type = “2..Topology”) for Topology Optimization (NX Nastran).

This manager can be used to create a *New* variable/region; *Edit, Delete*, *Copy*, or *Renumber* the variable/region currently highlighted in the *Available Optimization Relation(s)* list; highlight entities in the graphics window which are controlled by the variable/region currently highlighted in the list (*Show Selected*); or *Delete All* the variables/regions currently in the model.

Clicking the *New* button will open a “Relation” dialog box. The dialog box which appears depends on the current value of *Type*, which has four options: “0..Property”, “1..Material”, “2..Topology” and “3..Element”. Options 0, 1, and 3 are used for design optimization, while option 2 is used for topology optimization. Once a variable/region has been created using a specific *Type*, the dialog box will default to that type when displayed again in the same session of FEMAP. Once everything has been entered, click OK to return to the *Optimization Variable and Topology Region Manager* or click *More* to create another design variable. The various dialog boxes explained below.

**Type = “0..Property”**

Displays the *Property Relation* dialog box:




This dialog box is used to create a property design variable for design optimization. An *ID* and/or descriptive *Title* can be specified for the design variable, otherwise, the next available ID will be used and no title assigned.

In the *Options* section, select an existing property using the *Property* drop-down. While almost any property can be selected to create a “user-defined relation”, only Rod, Bar (Standard or NASTRAN cross-section), Beam (Standard or NASTRAN cross-section), Spring/Damper, DOF Spring, Plate, Spring/Damper to Ground, and DOF Spring properties will automatically populate both the *Type* field with a predefined name and *Name* drop-down with various property values which can be used as design variables. Check the *Edit* box to enter a different name in the *Type* field. The items which appear in the *Name* drop-down are different for each property type and correspond to the type of property currently selected in the *Property* drop-down. Each Property Optimization Variable writes a DESVAR entry with corresponding DVPREL1 entry.

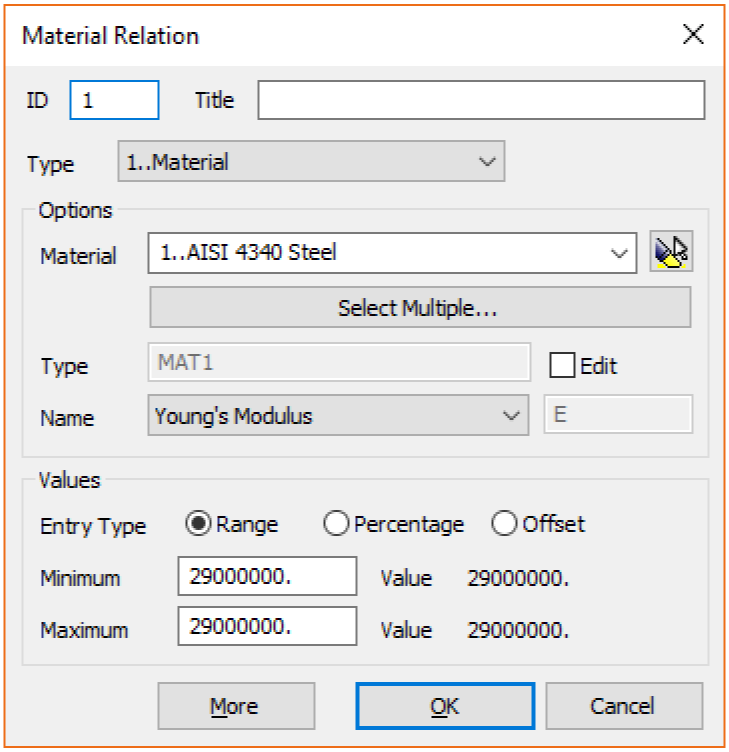
When using a property type which does not automatically populate the *Type* field and sets *Name* to “User Defined”, a name for *Type* and a variable name for *Name* must be manually entered. Even for property types which are supported to some level, a “User Defined” option will always appear at the bottom of the *Name* list. If selected, a variable name will need to be entered manually. In addition, only the Range option can be used when defining values for a “User Defined” design variable.

The *Values* section is where the *Minimum* and *Maximum* values are specified for the property design variable. These values can be enter as a *Range* (actual values, such as *Minimum* = 0.25, *Maximum* = 0.5), *Percentage* (for example, *Minimum* = 75%, *Maximum* = 100%), or *Offset* (for example, *Minimum* = -1, *Maximum* = 0.1 would allow a property values of 3.14 to range from 2.14 to 3.24).

The *Select Multiple* button in the options section can be used to select multiple properties in the model, but all the selected properties MUST be the same type, otherwise, FEMAP will remove some from the selection or ask for a new selection. Once property selection is complete, the Property drop-down will no longer be available and clicking OK will create multiple design variables, one for each property, using the specified *Type*, *Name*, and *Values*.

**Type = “1..Material”**

Displays the Material *Relation* dialog box:

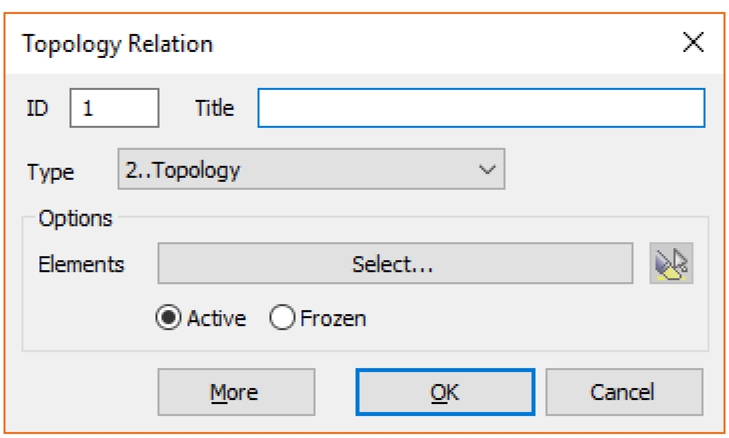



This dialog box is very similar to the *Property Relation* dialog box, with the only difference being it is used to define a Material design variable instead of a Property design variable. See Type = “0..Property” above for more information.

In the *Options* section, select an existing material using the *Material* drop-down. Again, while almost any material can be selected to create a “user-defined relation”, only Isotropic and Orthotropic (2D) Materials will automatically populate both the *Type* field with a predefined name and *Name* drop-down with various property values which can be used as design variables. Each Material Optimization Variable writes a DESVAR entry with corresponding DVMREL1 entry.

**Type = “2..Topology”**

Displays the *Topology Relation* dialog box:



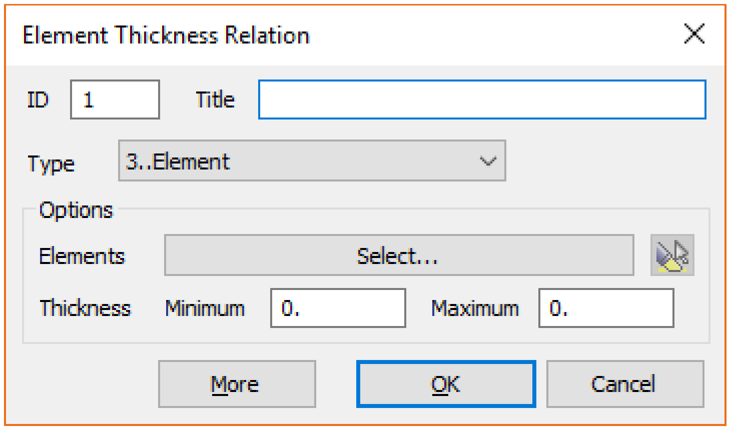

This dialog box is used to create a topology region for topology optimization. An *ID* and/or descriptive *Title* can be specified for the topology region, otherwise, the next available ID will be used and no title assigned.

Select *Active* or *Frozen* in the *Options* section to control if a topology region is considered “Active” (elements can be optimized) or “Frozen” (elements can not be optimized), then use the *Select* button to select elements using the standard entity selection dialog box. Use the *Preview* icon button to highlight the elements controlled by the region in the graphics window. Each Topology Region writes DVTREL1 entry with corresponding GROUP entry.

If only “Active” regions are selected for analysis, all other elements are “Frozen”. If only “Frozen” regions are selected for analysis, all other elements are “Active”. If a combination of “Active” and “Frozen” regions are selected for analysis, then only the elements in the “Active” regions are “Active”, all others are “Frozen”. Finally, any elements which reside in both an “Active” region and a “Frozen” region selected for analysis, will be “Frozen”.

**Type = “3..Element”**

Displays the *Element Thickness Relation* dialog box:

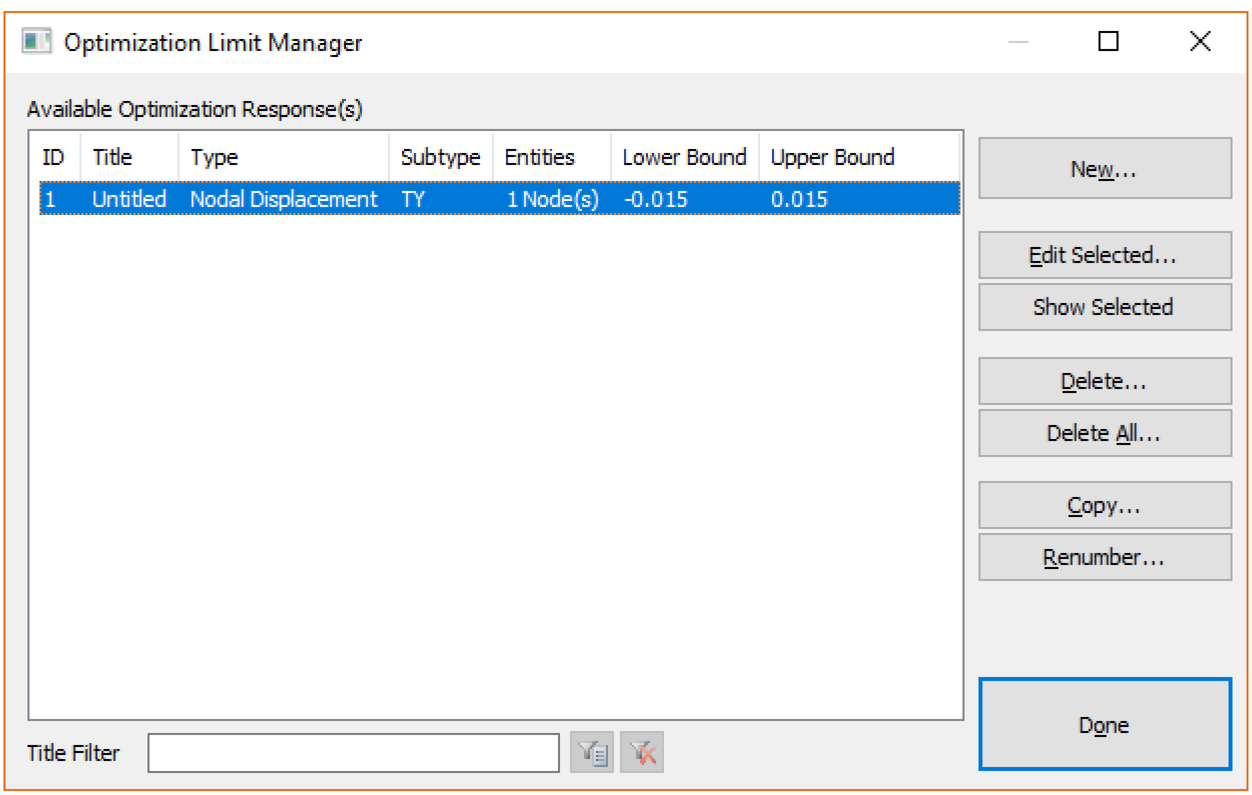



This dialog box is used to create a design variable for design optimization which varies thickness defined directly on plate elements, not defined by plate properties. An *ID* and/or descriptive *Title* can be specified for the design variable, otherwise, the next available ID will be used and no title assigned.

Use the *Select* button to select elements using the standard entity selection dialog box, then enter *Minimum* and *Maximum* values for *Thickness*. Use the *Preview* icon button to highlight the elements controlled by the design variable in the graphics window. Each Element Optimization Variable writes a DESVAR entry along with corresponding DVEREL1 and GROUP entries.

- Added *Model, Optimization, Limits* command which opens the *Optimization Limit Manager*, which is used to create, edit, copy, renumber, delete selected, or delete all Optimization Limits (responses).

Opens the *Optimization Limit Manager*:



This manager is used to create limits, or “responses”, for design optimization (NX Nastran or MSC Nastran) or Topology Optimization (NX Nastran).

This manager can be used to create a *New* response; *Edit, Delete*, *Copy*, or *Renumber* the response currently highlighted in the *Available Optimization Response(s)* list; highlight entities in the graphics window which are controlled by response currently highlighted in the list (*Show Selected*); or *Delete All* the responses currently in the model.

Clicking the *New* button will open a “Response” dialog box. The dialog box which appears depends on the current value of *Category*. Once everything has been entered, click OK to return to the *Optimization Limit Manager* or click *More* to create another response. Once a response has been created using a specific *Category*, the dialog box will default to that category when displayed again in the same session of FEMAP.

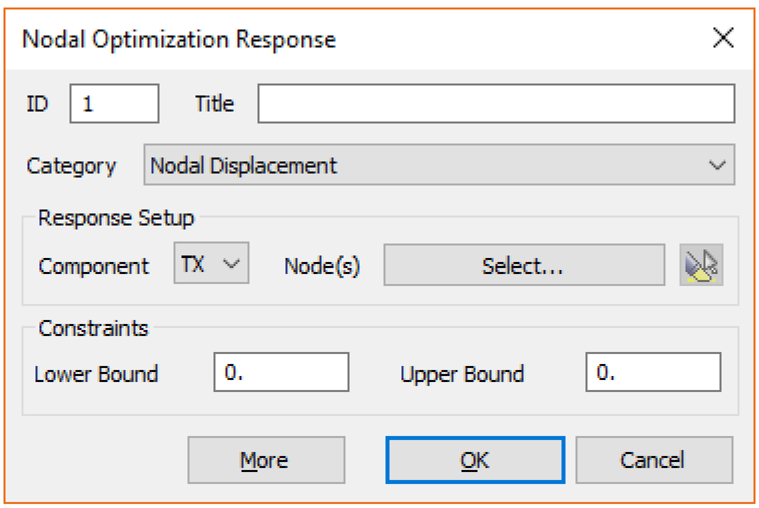
### *Category*

Defines the category of the response for design optimization or topology optimization, which has ten options: “Nodal Displacement”, “SPC Forces”, “Element Force, Stress, Strain”, “Strain Energy”, “Frequency”, “Eigenvalue”, “Buckling”, “Compliance”, “Volume”, and “Weight”. All options other than “Compliance” and “Volume” can be used for design optimization, while all options can be used for topology optimization.

In all cases, an *ID* and/or descriptive *Title* can be specified for the design variable, otherwise, the next available ID will be used and no title assigned. The various dialog boxes will be explained further below.

#### Nodal Displacement

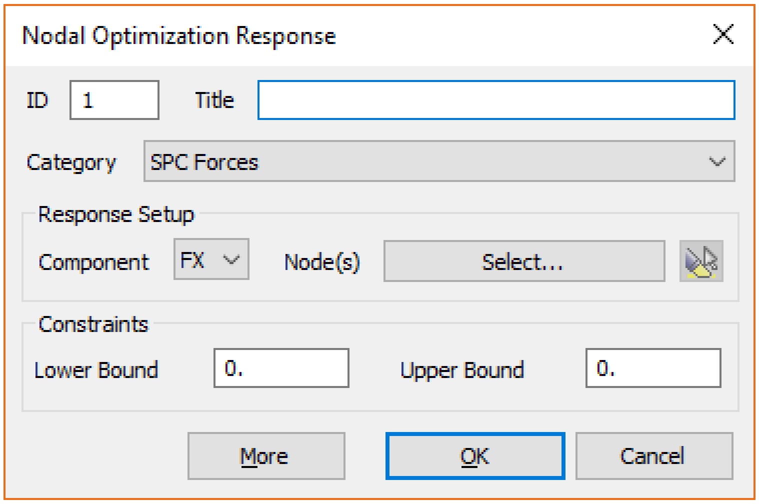
Displays the *Nodal Optimization Response* dialog box:




Used to limit nodal displacement. Use the *Component* drop-down in the *Response Setup* section to select a component of nodal displacement or rotation, then the *Select* button to select the nodes to limit displacement or rotation in the selected direction to the values specified for *Lower Bound* and *Upper Bound* in the *Constraints* section.

### SPC Forces

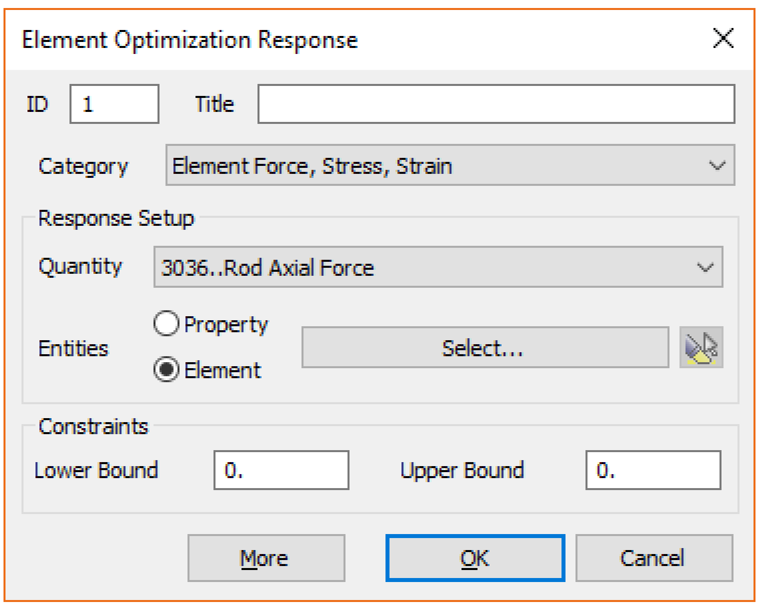
Displays the *Nodal Optimization Response* dialog box:




Used to limit nodal SPC forces. Use the *Component* drop-down in the *Response Setup* section to select a component of nodal SPC Force or Moment, then the *Select* button to select the nodes to limit the selected component of SPC Force or SPC Moment to the values specified for *Lower Bound* and *Upper Bound* in the *Constraints* section.

### Element Force, Stress, Strain

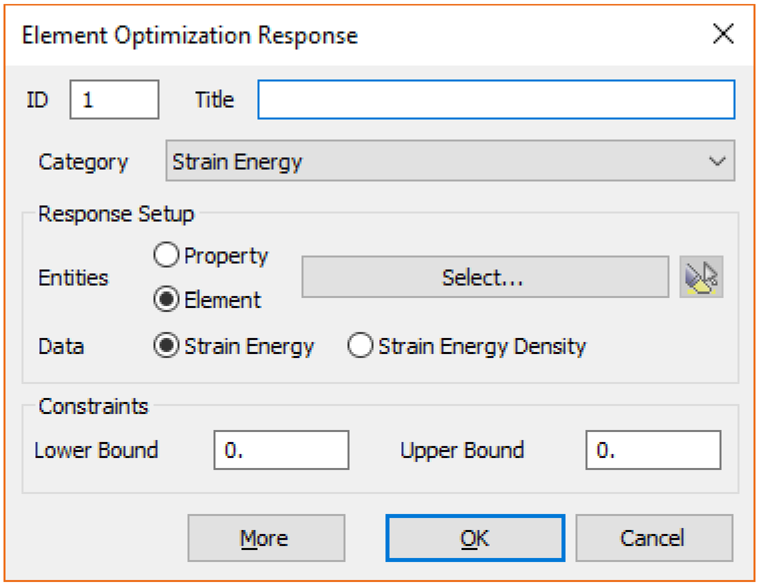
Displays the *Elemental Optimization Response* dialog box:




Used to limit a selected quantity of element force, stress, or strain. Use the *Quantity* drop-down in *Response Setup* section to select an element force, stress, or strain, then the *Select* button to select individual elements (*Element* option) or all the elements of the selected properties (*Property* option) to limit selected force, stress, or strain quantity to the values specified for *Lower Bound* and *Upper Bound* in the *Constraints* section.

**Strain Energy**

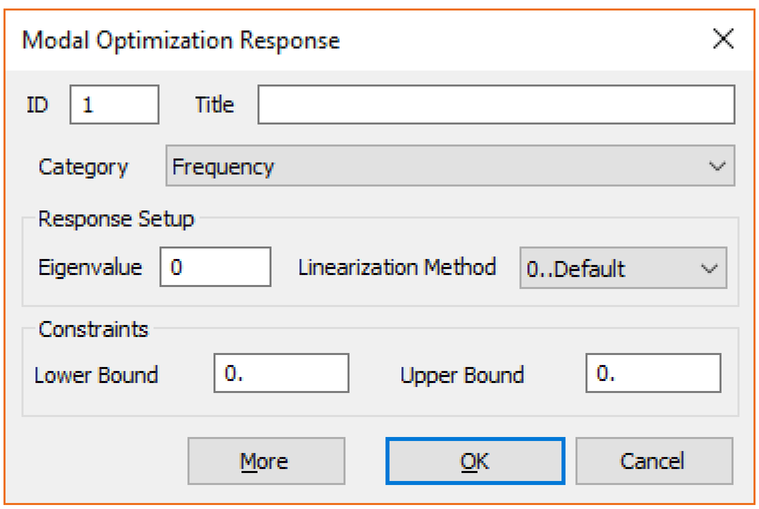
Displays the *Elemental Optimization Response* dialog box:




Used to limit Strain Energy or Strain Energy Density. Use the *Data* option in *Response Setup* section to choose either *Strain Energy* or *Strain Energy Density*, then the *Select* button to select individual elements (*Element* options) or all the elements of the selected properties (*Property* option) to limit strain energy or strain energy density to the values specified for *Lower Bound* and *Upper Bound* in the *Constraints* section.

**Frequency**

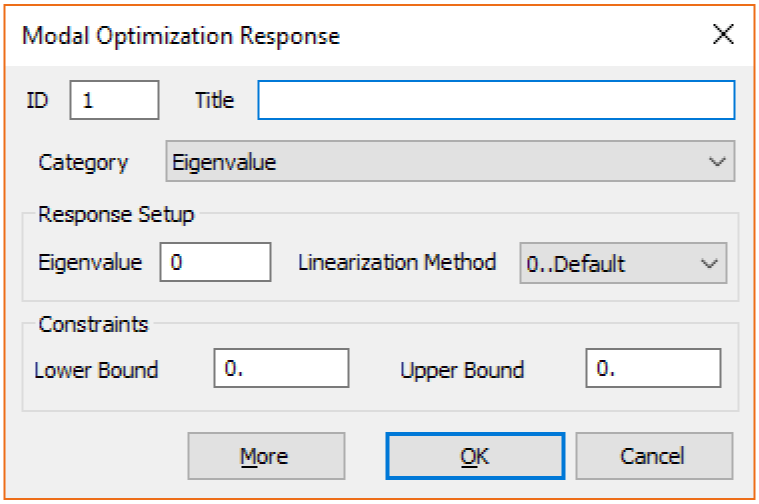
Displays the *Nodal Optimization Response* dialog box:




Used to limit Frequency for modal analysis. Use the *Eigenvalue* field to specify the “ID” of the mode, along with a *Linearization Method* (*Default*, *Direct*, or *Inverse*) to limit that mode to the frequency values defined by *Lower Bound* and *Upper Bound* in the *Constraints* section.

### Eigenvalue

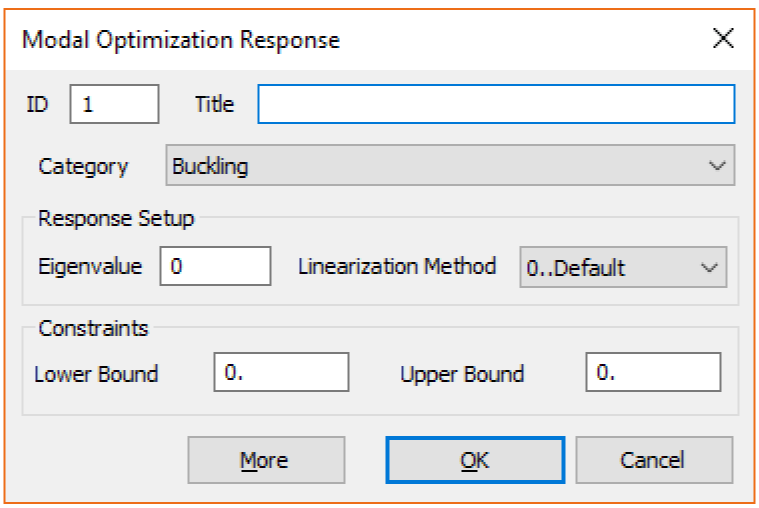
Displays the *Nodal Optimization Response* dialog box:




Used to limit Eigenvalue value for modal analysis. Use the *Eigenvalue* field to specify the "ID" of the mode, along with a *Linearization Method* (*Default*, *Direct*, or *Inverse*) to limit that mode to the eigenvalues values defined by *Lower Bound* and *Upper Bound* in the *Constraints* section.

### Buckling

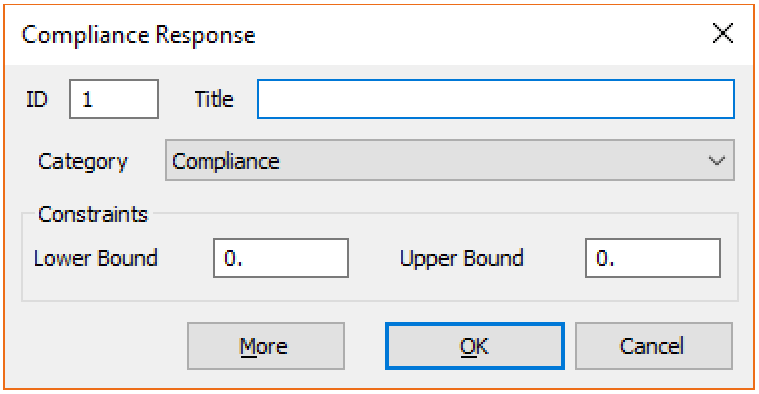
Displays the *Nodal Optimization Response* dialog box:




Used to limit the Frequency value for Buckling analysis. Use the *Eigenvalue* field to specify the "ID" of the mode, along with a *Linearization Method* (*Default*, *Direct*, or *Inverse*) to limit that mode to the frequency values defined by *Lower Bound* and *Upper Bound* in the *Constraints* section.

### Compliance

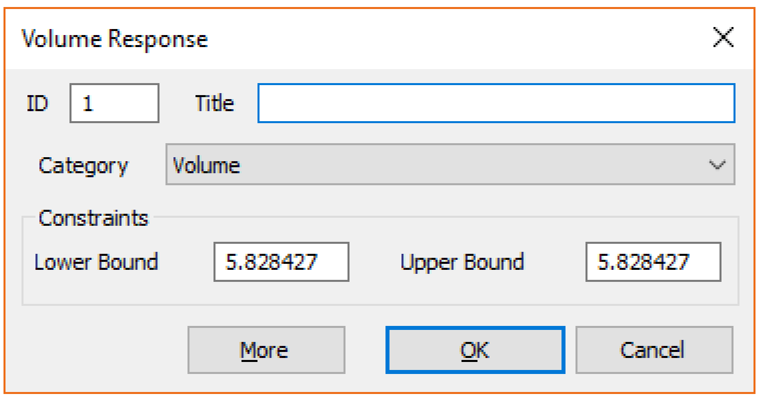
Displays the *Nodal Optimization Response* dialog box:




Used to limit the Compliance constraint for Topology Optimization to values defined by *Lower Bound* and *Upper Bound* in the *Constraints* section.

### Volume

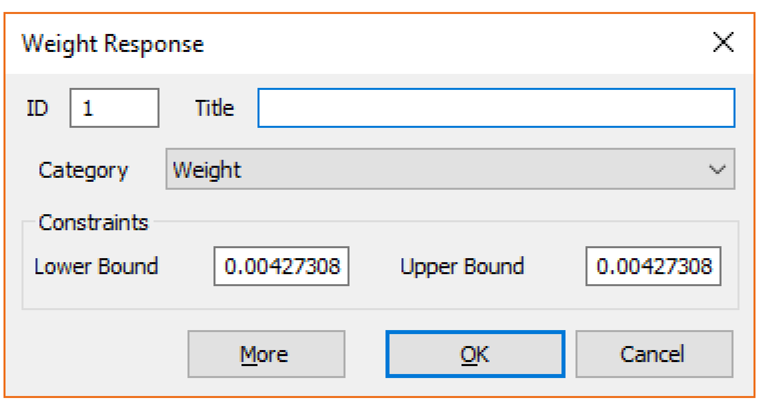
Displays the *Nodal Optimization Response* dialog box:




Used to limit the Volume constraint for topology optimizations to values defined by *Lower Bound* and *Upper Bound* in the *Constraints* section.

### Weight

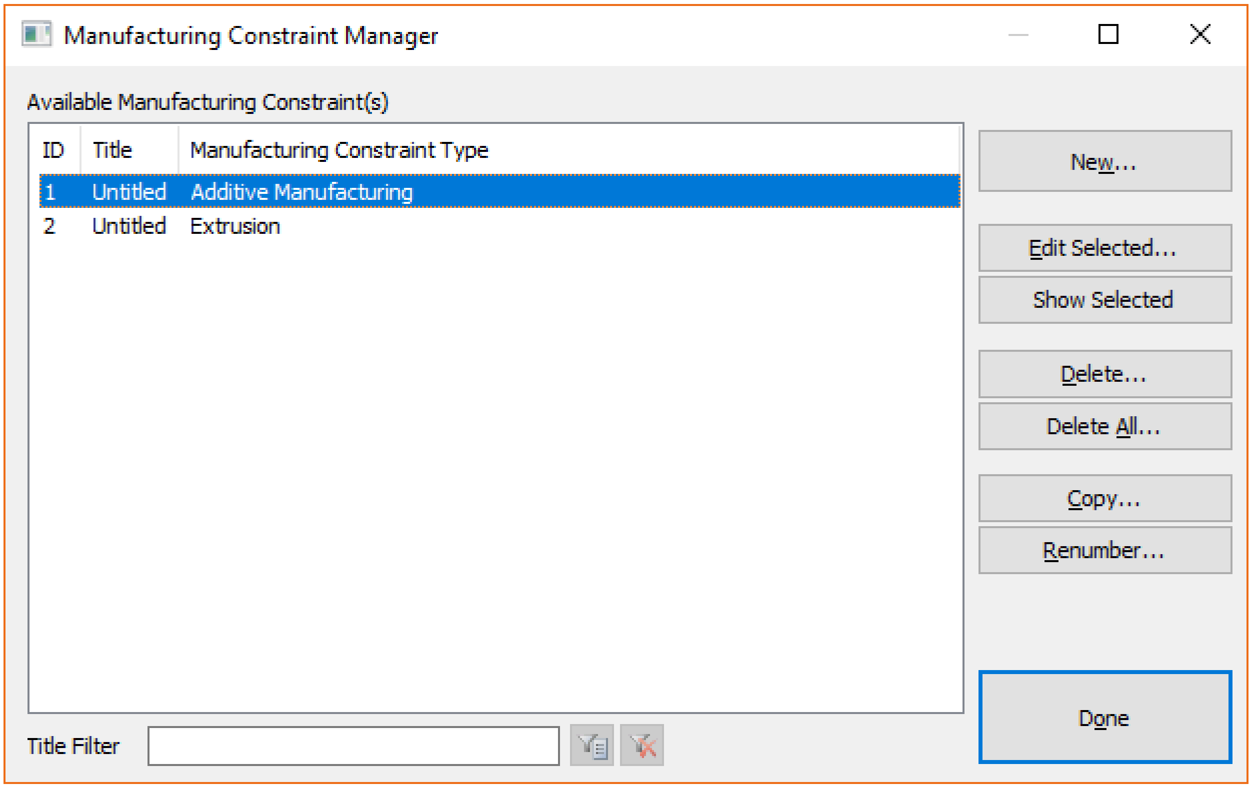
Displays the *Nodal Optimization Response* dialog box:




Used to limit the Weight constraint to values defined by *Lower Bound* and *Upper Bound* in the *Constraints* section.

- Added *Model, Optimization, Manufacturing Constraints* command which opens the *Manufacturing Constraint Manager*, which is used to create, edit, copy, renumber, delete selected, or delete all Manufacturing Constraints specified for Topology Optimization.

Opens the *Manufacturing Constraint Manager*:




This manager is used to create manufacturing constraints for Topology Optimization (NX Nastran).

This manager can be used to create a *New* manufacturing constraint; *Edit, Delete*, *Copy*, or *Renumber* the manufacturing constraint currently highlighted in the *Available Manufacturing constraint(s)* list; show a preview of any vector(s) or plane(s) specified for the manufacturing constraint highlighted in the list (*Show Selected*); or *Delete All* the manufacturing constraint currently in the model.

***Note:*** *Show Selected* will not be able to show anything in the graphics window for manufacturing constraints with *Type* set to “Checkerboarding Control”, “Maximum Size”, or “Minimum Size”, as these manufacturing constraints do not reference entities, vectors, or planes.

Clicking the *New* button will open a “Manufacturing Constraint” dialog box. The dialog box which appears depends on the current value of *Type.* Once everything has been entered, click OK to return to the *Manufacturing Constraint Manager* or click *More* to create another manufacturing constraint Once a manufacturing constraint has been created using a specific *Type*, the dialog box will default to that type when displayed again in the same session of FEMAP.

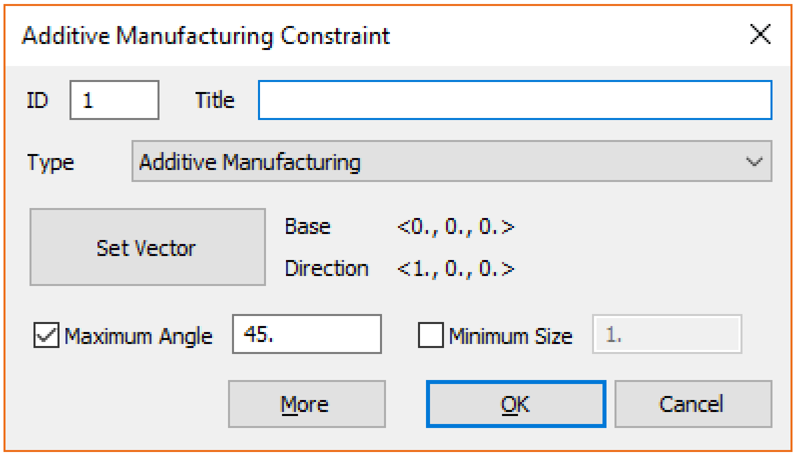
### *Type*

Defines the type of manufacturing constraint to enforce for topology optimization, which has eight options: “Additive manufacturing”, “Casting Die Direction”, Checkerboarding” Cyclic Symmetry”, “Extrusion”, “Minimum Size”, “Maximum Size”, and “Planar Symmetry”.

In all cases, an *ID* and/or descriptive *Title* can be specified for the design variable, otherwise, the next available ID will be used and no title assigned. The various dialog boxes will be explained further below.

### Additive Manufacturing

Displays the *Additive Manufacturing Constraint* dialog box:



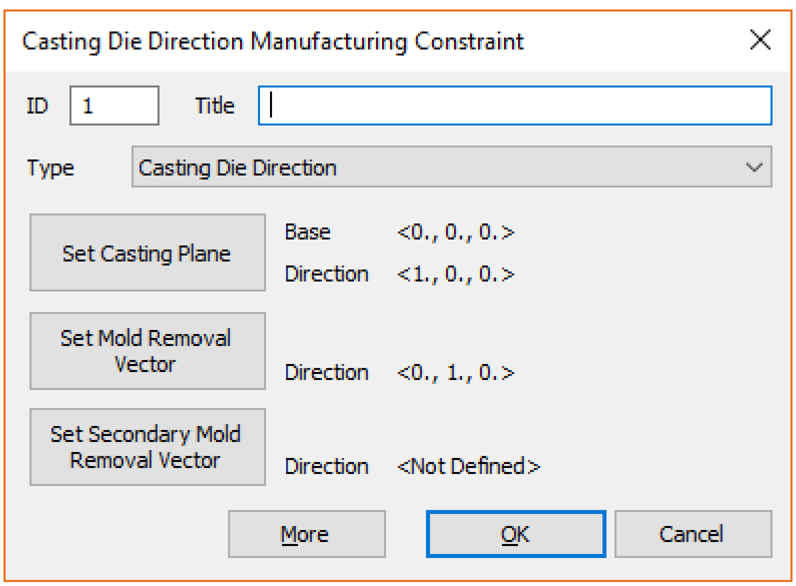

Used to enforce an Additive Manufacturing Constraint. The *Set Vector* button should be used to specify a vector where the base is on a plane for the “base plate” and the tip is normal to the base plate in the direction of material addition. *Maximum Angle* is the maximum angle from the normal that points away from the base plate, for overhangs, in degrees, while *Minimum Size* is the minimum allowed size. Both *Maximum Angle* and *Minimum Size* can be disabled, indicating there is no such constraint. Writes a DMNCOM bulk entry with TYPE=ADDM.

Additive manufacturing builds material layer by layer. The previous layers in a build-up must be sufficient to support consecutive layers. When building up overhanging geometry, if the overhang is built in an aggressive angle, there is a possibility that the structure can collapse as a result of cantilevered material. You define a maximum overhang angle and/or a minimum size applicable to any slender members, as well as the manufacturing direction.

Overhang angles are measured relative to the positive manufacturing direction vector.

### Casting Die Direction

Displays the *Casting Die Direction Manufacturing Constraint* dialog box:

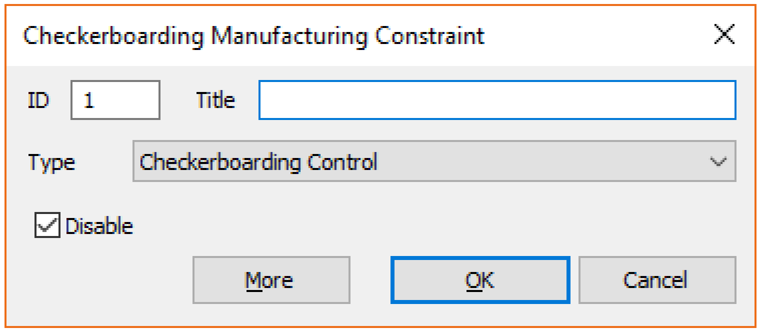



Used to enforce an Casting Die Direction Manufacturing Constraint. The *Set Casting Plane* button should be used to specify a vector with the base on the “casting plane” and the tip being normal to the casting plane. The *Set Mold Removal Vector* should be used to specify a vector that defines the first direction of “mold removal”, while the *Set Secondary Mold Removal Vector* should be used to specify a vector that defines the second direction of mold removal. If *Set Secondary Mold Removal Vector* is not defined, the only the first direction is used. Writes a DMNCOM bulk entry with TYPE=CDID.

Parts that are cast should not have any pockets that cannot be formed or protrusions that would interfere with the mold pieces from parting, or with the part coming out of the mold. You define the coordinates of a point on the casting plane, a vector normal to the casting plane, a vector along which one part of the mold travels when separating, and another, optional, vector along which the other part of the mold, if it exists, travels when separating.

### Checkerboarding Control

Displays the *Checkerboarding Control Manufacturing Constraint* dialog box:

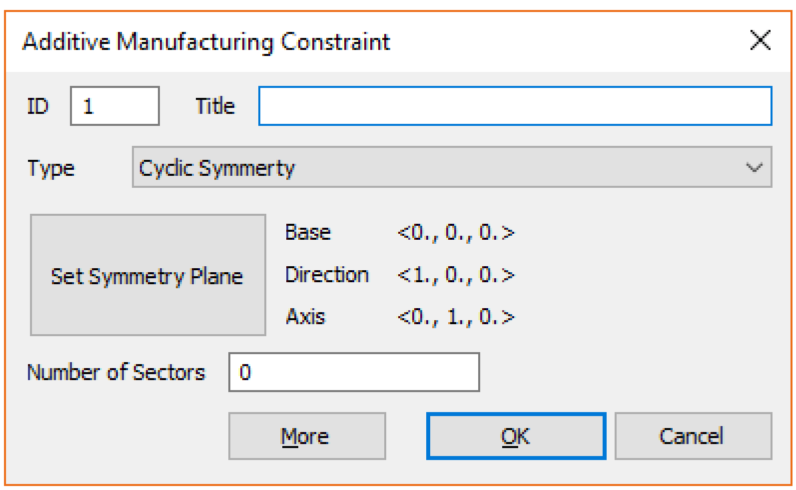



Used to enforce an Checkerboarding Manufacturing Constraint. The checker-boarding control manufacturing constraint is on by default, even when the DMNCON bulk entry has not been defined. Simply leave the *Disable* check box checked to disable checker-boarding control, as this manufacturing constraint will define a DMNCON bulk entry with TYPE= CHBC and a negative real number in the OFF-FLAG field.

Checker-boarding is a condition where topology optimizers remove material in an alternating pattern similar to a checker board, when simpler finite elements are used. It is undesirable because it does not represent an optimal distribution of material and the results are difficult to manufacture. The checker-boarding control constraint can be used to help prevent this condition from occurring.

### Cyclic Symmetry

Displays the *Cyclic Symmetry Manufacturing Constraint* dialog box:



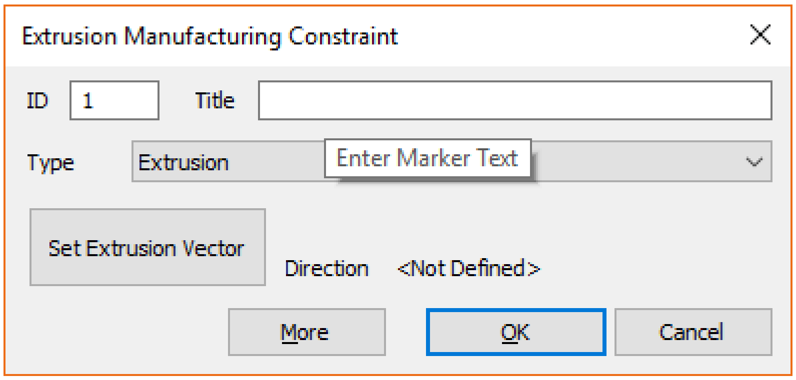

Used to enforce an Cyclic Symmetry Manufacturing Constraint. The *Set Symmetry Plane* button should be used to specify a plane with the base on the “symmetry plane”, point 1 defining the axis of rotation, and point 2 defining a vector perpendicular to the axis of rotation, along one of the edges of the model on the 0-degree symmetry plane. *Number of Sectors* is the number of sector which will fill 360 degrees. When *Number of Sectors* is a positive integer, the symmetry is that of repeated sectors. When *Number of Sections* is a negative integer, the symmetry is that of reflected sectors. Writes a DMNCOM bulk entry with TYPE=SYMC.

You provide a full circular model which has repeated cyclic symmetry sectors. The mesh on each sector does not need to match. NX Nastran will work to symmetrize the normalized mass density (NMD) values on each sector. You define the axis of rotation, the number of sectors (NSECT) within 360 degrees, and relative to a 0-degree symmetry plane, you define a point on that symmetry plane and a radial vector perpendicular to the axis of rotation along one of the edges of the symmetry plane. NX Nastran sweeps the 0-degree symmetry plane counter clockwise by the angle = 360 / | NSECT | to determine the consecutive symmetry planes.

When NSECT is a positive integer, NX Nastran treats your model as having repeated sectors. When NSECT is a negative integer, NX Nastran treats your model as having reflected sectors. With repeated symmetry, each sector is similar to every other sector. With reflected symmetry, every sector is the mirror image of the previous sector. You can use reflected symmetry only with an even number of sectors.

### Extrusion

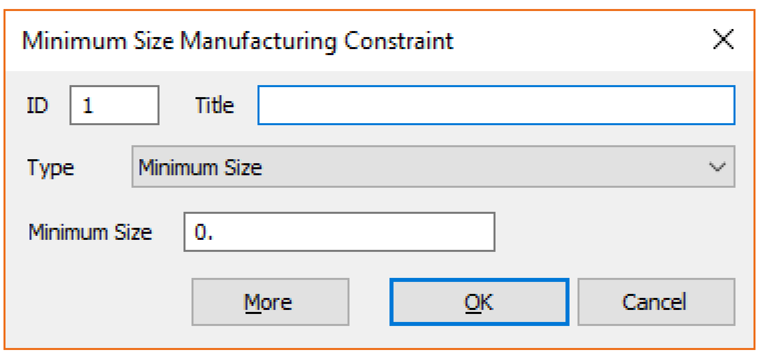
Displays the *Extrusion Manufacturing Control* dialog box:




Used to enforce an Extrusion Manufacturing Constraint. The *Set Extrusion Vector* button should be used to specify a vector defining the extrusion direction. Writes a DMNCOM bulk entry with TYPE=EXTC. Extruded parts must have material continuity in the extrusion direction. You define the straight line extrusion direction.

### Minimum Size

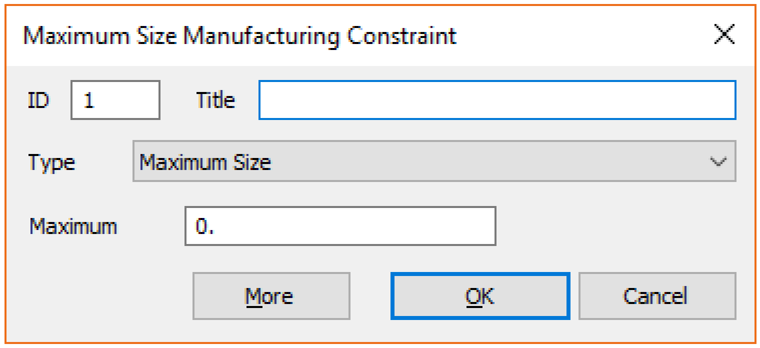
Displays the *Minimum Size Manufacturing Control* dialog box:




Used to enforce an Minimum Size Manufacturing Constraint. Simply enter a value in the *Minimum Size* field. Writes a DMNCOM bulk entry with TYPE=MINS. This allows you to control the minimum member size for created “slender” members. You define the minimum cross-sectional “thickness” size.

### Maximum Size

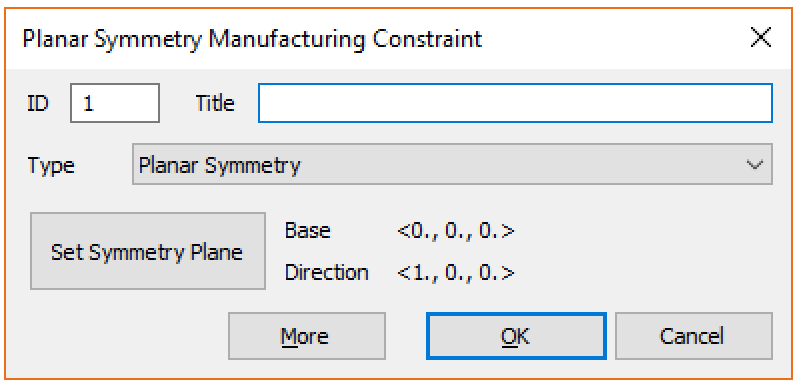
Displays the *Maximum Size Manufacturing Control* dialog box:




Used to enforce an Maximum Size Manufacturing Constraint. Simply enter a value in the Maximum Size field. Writes a DMNCOM bulk entry with TYPE=MAXS. This allows you to control the maximum member size. For example, if you define the maximum member size, truss members created by the optimization process will not be any “thicker” than the specified size.

### Planar Symmetry

Displays the *Planar Symmetry Manufacturing Constraint* dialog box:




Used to enforce an Planar Symmetry Manufacturing Constraint. The *Set Symmetry* button should be used to specify a vector where the base is on the “symmetry plane” and the tip is normal to the symmetry plane. Writes a DMN-COM bulk entry with TYPE=SYMP.

With this condition, your model must be meshed on both sides of the symmetry plane. The mesh on each side of the symmetry plane does not need to match. NX Nastran works to symmetrize the NMD values on both sides of a given symmetry plane. You define a point on the symmetry plane and a vector normal to the symmetry plane.

## Listing

- Added *List, Geometry, Mesh Point* command to list details about mesh points to the Messaged pane and any other destination(s) specified by the *List, Destination* command.
- Updated *List, Output, Results to Data Table* command by adding *Show Summary Table* option to the *Send Results to Data Table* dialog box, which adds a summary table to the bottom of the *Data Table* and contains Max and Min Values, their corresponding IDs, Sum of the Values, and the Average Value for each column.
- Updated *List, Output, Results to Data Table* command by adding *Transform* section to the *Send Results to Data Table* dialog box. This section can be used to send transformed output data to the *Data Table* using the options specified for the active view (*Active View* option) or specified via the *Transform...* button (*Custom...* option).
- Updated *List, Output, Results Ranking to Data Table* command by adding *Transform* button to the *Rank Output to Data Table* dialog box. This section can be used to send ranked output data to the *Data Table*, which has been transformed using the options specified via the *Transform...* button.
- Updated *List, Output, Contoured Results to Data Table* command by adding a header to the to of that *Data Table*, which describes the selected output sets/output vectors in the *Data Table*, including transformations.

## Groups and Layers

- Added the *Group, Operations, Generate From Criteria* command, which is only available when a Criteria plot is being displayed in the graphics window (i.e., *Contour Style* is set to *Criteria* in the *View, Select* command or *Criteria* is selected from the *Style* drop-down in the *Contour Tool* of the *PostProcessing Toolbox*). Creates a group containing all elements which fulfill the criteria options currently set for the active view, which may be different than all visible elements, especially if there are multiple element types in the model.
- Added the *Group, Mesh Point* submenu, which allows you to define, edit, and delete the rules that will be used to select mesh points. You can select mesh points into your group based on their *ID, Color, Layer*, associated with geometric entities (*on Point*, *on Curve*, *on Surface*, *in Solid*), or any combination of these methods.
- Improved performance of the “Add Related Entities” capability of Groups, which can be done using various methods throughout FEMAP. Previously in models that had large numbers of geometry-based contact, along with large numbers of solid elements, this command could take a long time to complete. This also improved performance of other commands and tools which use this capability, such as the *Entity Locator* found in the *Meshing Toolbox*.

## Views

- Added *View, Autoscale, Fit* command, which is similar to the *View, Autoscale* command. It too bases its calculations on the overall model dimensions. However, this command only considers the current orientation of the model. It projects the overall dimensions into the current view, then adjusts the magnification factor to attempt to fill the screen with the visible entities. This will always result in a larger image than *View, Autoscale*. If the model geometry is non-rectangular, or has cutouts, this option still might not fill the view. Also, this command will automatically adjust the centering of the model.
- Added “Mesh Point” option in “Labels, Entities and Color” Category of *View, Options* command, which controls visibility, label mode, and color mode for mesh points.
- Added “Element - Cohesive” option in “Labels, Entities and Color” Category of *View, Options* command, which controls visibility of property thickness, label mode, and color mode for mesh points.
- Added “Element - Shell” option in “Labels, Entities and Color” Category of *View, Options* command, which can be used to suppress thick edges of shell elements to improve performance and aid transparency.
- Added “2..DOF - Shrink Always” option to *Release Labels* section for “Element - Offsets/Releases” option in “Labels, Entities and Color” Category of *View, Options* command, which can be used to always display the beam release degrees of freedom away from the End A and End B nodes.
- Added “Local Components” options to *Color/Component* section for “Load - Force and Bearing”, “Load - Moment and Torque”, “Load - Accleration”, “Load - Velocity”, and “Load - Enforced Displacement” options in “Labels, Entities and Color” Category of *View, Options* command, which can be used to display these load types as components in the coordinate system in which they are defined instead of as “Global Components”.
- Added “Feature Line” option in “Tools and View Style” Category of *View, Options* command, which controls if “Feature Lines” should be displayed on the mesh.

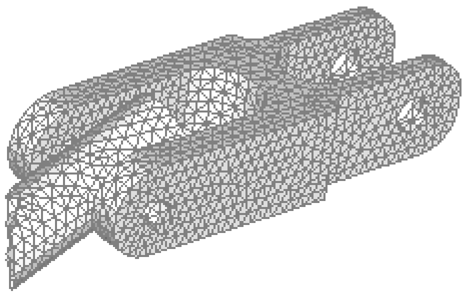
When *Draw Entity* is enabled, attempts to create “feature lines” on a mesh which essentially creates an “outline” around the different parts in the model. Where the feature lines appear is controlled via a combination of the option selected in the *Element Control and Silhouette* section and the value specified in the *Angle* field, which is used as a “break angle” (i.e., if the angle between elements is above this value, create a feature line between the elements).

When *Element Control and Silhouette* is set to “0..All Elements” or “1..All Elements, Silhouette” all of the elements in the model will be used to generate the feature lines, whether they are visible or not. When set to “2..Drawn Elements” or “3..Drawn Elements, Silhouette”, only the elements which are currently visible will be considered. When using either of the “Silhouette” options, additional features lines will be drawn to create an outline around the model using the current orientation and will change as the model is rotated.

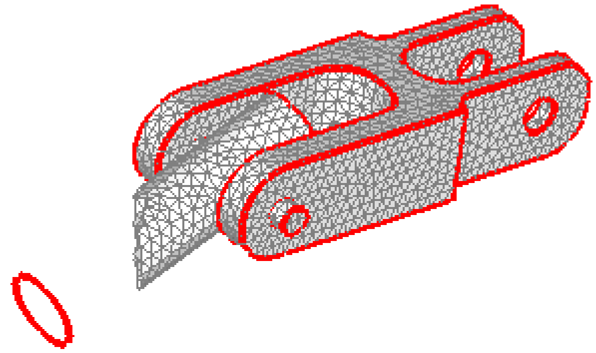
The *Surface Division* section can be used to create additional feature lines between elements which reference different properties (“1..Property”) or different materials (“2..Material”). By default, this option is set to “0..None”.

The default value for *Angle*, which is “5”, may not create the desired feature lines. Consider changing the value, especially when using this option for a model which has been meshed with tetrahedral elements

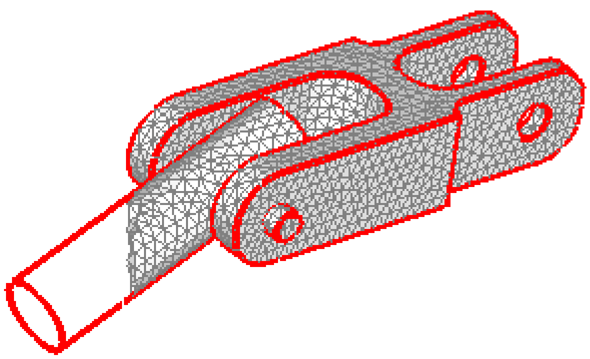
Model with Feature Line turned Off



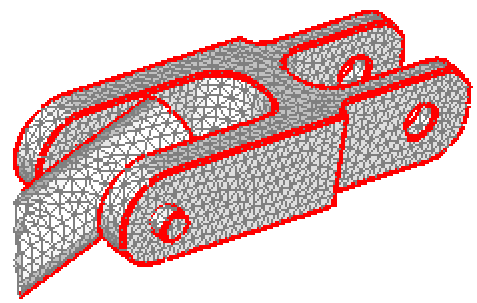
Using “”0..All Elements” option



Using “1..All Elements, Silhouette” option

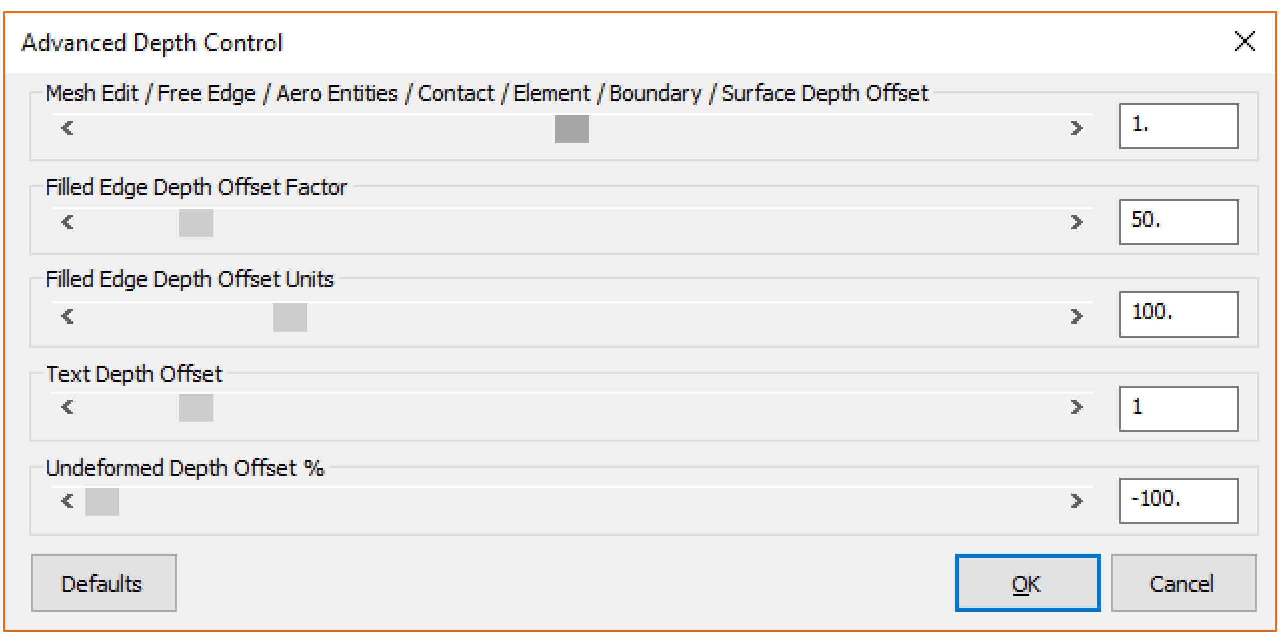


Using “3..Drawn Elements, Silhouette” option



- Added *Advanced Depth Control* button to “Graphics Options” option in “Tools and View Style” Category of *View, Options* command.

Clicking the *Advanced Depth Control* button will open the *Advanced Depth Control* dialog box:




This dialog box contains five different slider bars, each of which can be used to modify the value of an option that controls depth and/or offset of a type of entity. The values for four of these options can be specified elsewhere in *View Options* dialog box, which is detailed below. Using to slider bar to update a value should update how items are displayed in the graphics window, providing immediate feedback. Changing the value for any of these options also changes the value in the other location in the *View Options* dialog box.

The option at the top of the dialog box, *Mesh Edit/Free Edge/Aero Entities/Contact/Element/Boundary/Surface Depth Offset*, can only be specified with this control. This option controls the spacing between different view modes and entity types. Essentially, items are “stacked” from closest to the user to furthest away from the user in the following order: Dynamic Mesh Editing (i.e., using the *Mesh Locate* tool in the *Meshing Toolbox*), plot of Free Edges, Aero Control Surfaces, Aero Splines, Aero Bodies/Panels, Connection and other Regions, Elements, Boundary Surfaces, and Surfaces. For example, if a surface is meshed, the elements should be visible, not the surfaces, but when a highly curved surface with a coarse mesh, this will not always be true. If it is desired to have the elements on meshed surfaces always attempt to appear in front of the surface associated to the mesh, move the slider bar to the right to increase this value.

*Filled Edge Depth Offset Factor* and *Filled Edge Depth Offset Units* correspond to the *Offset Factor* and *Offset Units* values described above in this section.

*Text Depth Offset* corresponds to the *Text Depth Offset* value described in "Label Parameters"

*Undeformed Depth Offset* corresponds to the value for *Depth Offset %* in Section 8.3.6, “Undeformed Model...”.

- Added *Discrete Values* capability to the *View, Advanced Post, Contour Model Data* command.

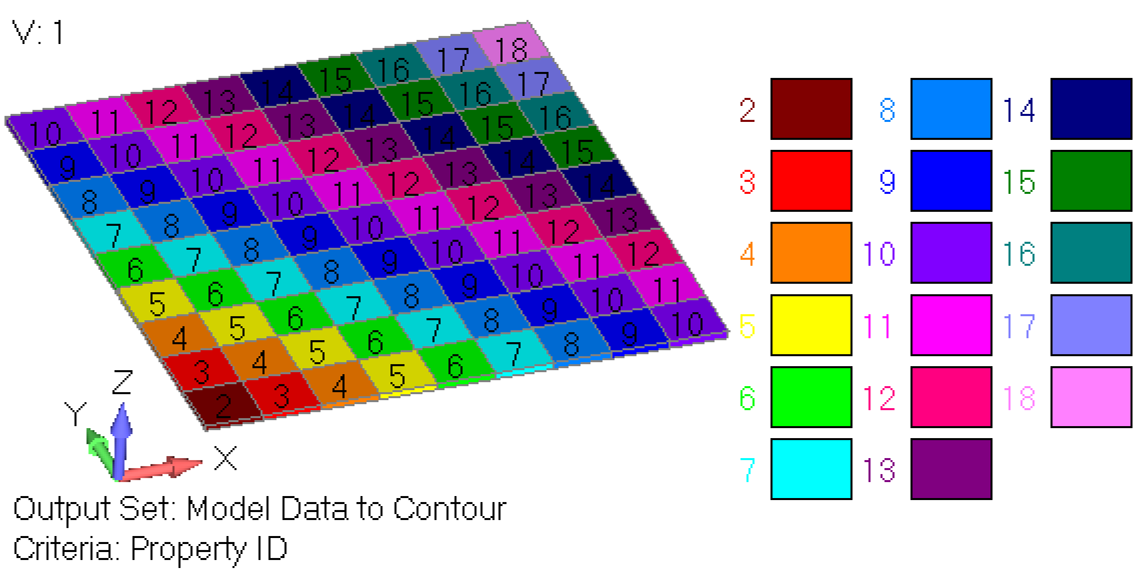
### Discrete Values

To display a “Discrete Value Plot”, enable *On* in this section, then select an existing “Discrete Value Set” using the drop-down. See "Discrete Value Sets" for more information about creating, editing, and managing Discrete Value Sets. When enabled, the selected Discrete Value Set will be used to display values for the item currently highlighted in the *Contour Data From* section, along with the options specified in the *Show As* and *Show On* sections.

While a Discrete Value Plot looks similar to a Criteria plot, it differs in several ways. First, the number of “levels” displayed, along with the color assigned to each level, is determined from the selected Discrete Value Set. Second, the *Contour/Criteria Legend* appears as individual “color blocks” with the value or range of values associated to each block to the right or the left of the block. Finally, it is possible for the color blocks to appear in multiple columns when the *Contour/Criteria Legend* is oriented vertically, or multiple rows when oriented horizontally.

***Note:*** If using a Discrete Value Set created using the *From Data* method, be sure the corresponding item is selected in the *Contour Data From* section, otherwise, the plot may not be useful. For example, if the Discrete Value Set was created with *Data Type* set to “Property ID”, make sure the “Property” option in the “Entity ID” branch of the tree structure is selected in the *Contour Data From* section.

For example, here is a Discrete Value Plot of “Property ID”, *Show As* set to *Criteria* and *Allow Labels* enabled:



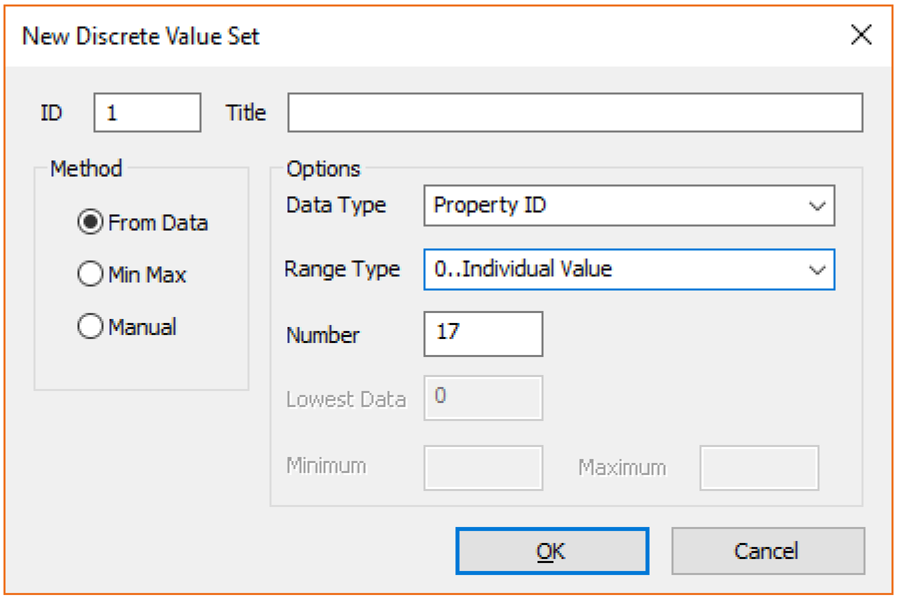
### Discrete Value Sets

Three buttons related to Discrete Value Sets, *New*, *Edit*, and *Manage*, are available in the *Discrete Values* section. The *New* button is used to quickly create a new Discrete Value Set using the *New Discrete Value Set* dialog box, which will be explained later in this section, while the *Edit* button can be used to edit the Discrete Value Set currently selected in the drop-down using the *Discrete Value Set* dialog box, also explained later in this section.

The *Manage* button will display the *Discrete Value Set Manager*, which can be used to create a *New* set using the *New Discrete Value Set* dialog box; *Edit*, *Renumber*, *Delete*, or *Copy* the set currently highlighted in the list found in the *Discrete Value Set Manager*; or *Delete All* sets which exist in the model.

### New Discrete Value Set dialog box

The *New Discrete Value Set* dialog box is used to create a New Discrete Value Set:




A specific *ID* and/or descriptive *Title* may be entered, but are not required, as the next available ID and a automatic title generated from the items specified in the Options section are the default.

If an item is currently highlighted in the *Contour Data From* section, then *Method* will automatically be set to *From Data, Data Type* set to the item in *Contour Data From*, and *Range Type* set to “0..Individual Value” with *Number* set to the appropriate number of discrete values for the *Data Type*, if that can be determined, or set to “1..Range by equal value” with *Number* set to 10. In many cases, this is all that will be needed to create an appropriate Discrete Value Set for the selected *Data Type*. Otherwise, it will be required to specify options and values manually.

### Method and Options

The selected *Method* controls what is available in the *Options* section to create a valid Discrete Value Set, which are then used to create the appropriate number of rows and columns in the *Discrete Value Set* dialog box.

***From Data*** - based on selected *Data Type*, attempts to determine appropriate *Range Type* and *Number* to create a useful Discrete Value Set. The *Range Type* and/or *Number* can be modified to create the desired number of levels and any *Range Type* can be used for any *Data Type*, although some may not be as useful as the default option.

0..Individual Value - creates a single row of individual IDs or Values. Only elements which have an ID or value which matches an ID or Value in the list will have a color assigned. This is useful when creating plots of Property or Material IDs or when each unique value should be displayed, like thickness of plate properties.

1..Range by equal value - for IDs, takes Highest ID - Lowest ID, then divides them by the value specified for *Number*, then rounds down to the closet integer value. This integer value is used for all levels except the final level, which may be larger. For values, takes “Maximum value for the Data Type” - “Minimum value for the Data Type”, then divides the difference by the value specified for *Number*. This is useful to display values which are potentially similar to one another as a single color, especially when a large number of values exist, but it is only important to know which portions of the model have similar values to one another.

2..Range by equal number - attempts to create ranges of IDs or values which have an equal number of elements in each range. Depending on the item selected in the *Data Type* drop-down, this may not be the best option, especially if displaying a quantity which may have many similar values, such as element quality.

3..Range by largest gap - finds the largest gap between IDs in a set of IDs or between Values in a set of values, then continues to find the largest remaining gaps in the set of IDs or set of values until the number of levels equals the value specified for *Number*. For instance, elements have IDs of 1-10, 101-110, 151-160, and 166-170. If *Number* is set to 2, then the largest gap is between element 10 and element 101, so elements 1-10 would be in “Level 1”, while all other elements would be in “Level 2”. If *Number* is set to 3, then the next largest gap is between 110 and 151, so elements 1-10 would be in “Level 1”, elements 101-110 would be in “Level 2” and the remaining elements would be in “Level 3”.

This table indicates which columns will be created in the *Discrete Value Set* dialog box when using the various options available for *Range Type* when *Method* is set to *From Data*:

| Range Type | Column 1 | Column 2 | Column 3 |
|---|---|---|---|
| 0..Individual Value | ID/Value | Color | N/A |
| 1..Range by equal value | Lower ID/Value | Upper ID/Value | Color |
| 2..Range by equal number | Lower ID/Value | Upper ID/Value | Color |
| 3..Range by largest gap | Lower ID/Value | Upper ID/Value | Color |

***Min Max*** - offers two options for *Data Type*, “0..Integer” or “1..Float” (i.e., real number). *Number* specifies the number of “levels” (i.e., equal divisions) to create between the values specified for *Minimum* and *Maximum*.

This table indicates which columns will be created in the *Discrete Value Set* dialog box when using the various options available for *Data Type* when *Method* is set to *Min Max*:

| Data Type | Column 1 | Column 2 | Column 3 |
|---|---|---|---|
| 0..Integer | Lower ID (Default = 0) | Upper ID (Default =1000) | Color |
| 1..Float | Lower Value (Default = 0.0) | Upper Value (Default = 1.0) | Color |

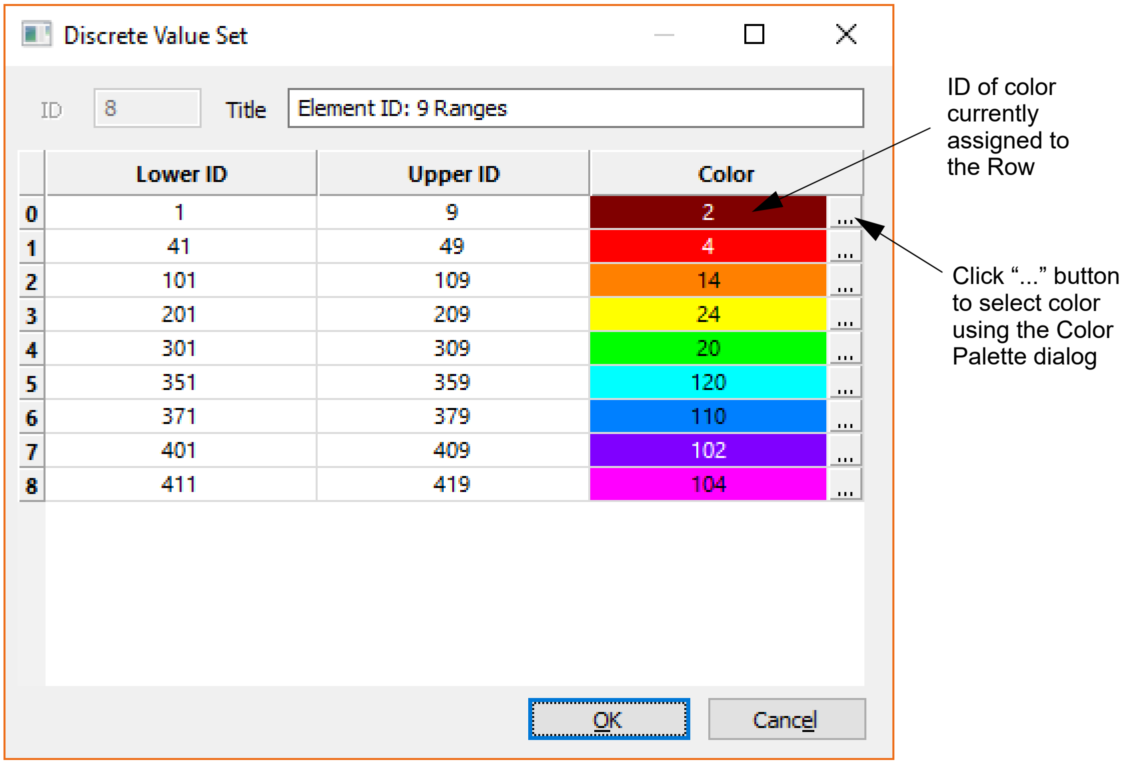
***Manual*** - offers two options for *Data Type*, “0..Integer” and 1..Float” (i.e., real number), and two options for *Range Type*, “0..Individual Value” and “1..Range by equal value”, which can be used in any combination. Since no additional information can be entered, these selections simply specify the number of columns which will be created for a single row in the *Discrete Value Set* dialog box, where additional rows can be added, then data entered.

This table indicates which columns will be created in the *Discrete Value Set* dialog box when using the various options available for *Data Type* and *Range Type* when *Method* is set to *Manual*:

| Data Type | Range Type | Column 1 | Column 2 | Column 3 |
|---|---|---|---|---|
| 0..Integer | 0..Individual Value | ID | Color | N/A |
| 0..Integer | 1..Range by equal value | Lower ID | Upper ID | Color |
| 1..Float | 0..Individual Value | Value | Color | N/A |
| 1..Float | 1..Range by equal value | Lower Value | Upper Value | Color |

### Discrete Value Set dialog box

The *Discrete Value Set* dialog box is used to update existing data or add new rows to a Discrete Value Set:




It is possible to change the current title by entering new text into the *Title* field.

When the column header says *ID*, *Lower ID*, or *Upper ID*, the column is expecting integers and will simply round any real numbers to the nearest whole number once OK has been clicked to save the Discrete Value Set. When the column header says *Value*, *Lower Value*, or *Upper Value*, the column is expecting real numbers.

The column furthest to the right will always be *Color*, which displays the ID of the color in a cell highlighted with the selected color. To change the color for a row (i.e., "level"), click in the cell and enter the ID of a color manually or click the "..." icon button to select a color from the *Color Palette*. See Section 4.3.5, "Color Palette" in the FEMAP User Guide for more information.

### *Context Sensitive Menus (All Functions/Tables)*

There is a context sensitive menu which will appear when cell(s) are highlighted in the *Discrete Value Set* dialog box and the right mouse button is clicked. This menu allows you to:

**Copy Selected** - copies the values in the highlighted group of cells to the clipboard.

**Paste** - pastes values from the clipboard into the highlighted cells.

**Clear Selected** - clears the value(s) in the highlighted cell(s).

The context sensitive menu for column headers includes the *Clear Selected* command along with:

**Auto Column Width** - automatically sets the column width to make all characters visible in all cells.

**Column Width** - allows specification of width, in pixels, for any number of selected columns.

The context sensitive menu for row headers also includes the *Clear Selected* command along with these other options:

**Add Rows** - adds the number of specified rows below the last row currently in *Discrete Value Set* dialog box.

**Insert Row** - inserts a new row beneath the current row in *Discrete Value Set* dialog box.

**Delete Row**- removes the current row from *Discrete Value Set* dialog box.

**Split Row** - only available when a single row is selected in *Discrete Value Set* dialog box, and allows the selected row to be split into two rows using the value specified in the *Split range at value* dialog box.

**Combine Rows** - only available when multiple rows are selected in the *Discrete Value Set* dialog box, and allows the values of any number of selected rows to be combined into a single row.

**Row Height** - allows specification of height, in pixels, for all rows in *Discrete Value Set* dialog box.

Additional commands may be available on the column header and row header context-sensitive menus when a certain type of table is loaded in the *Function/Table Editor.*

## Output and Post-Processing

- Added *View, Advanced Post, Dynamic Criteria* command. This functionality can also be accessed by using the *Dynamic Control* button when *Style* is set to *Criteria* in the *Contour* tool of the *PostProcessing Toolbox*.

Allows you to dynamically update the *Limits Mode* option and *Maximum* and/or *Minimum* values to use as criteria limits in the model. When you select this command, you will see the *Dynamic Criteria Control* box:


#### Limits Mode

The following table lists the available modes and their uses:

| Limits Mode | Minimum | Maximum | Result |
|---|---|---|---|
| No Limits | - | - | No Criteria. All elements pass. |
| Above Maximum | - | Yes | Elements with output values greater than Maximum pass. |
| Below Minimum | Yes | - | Elements with output values less than Minimum pass. |
| Between | Yes | Yes | Elements with output values between Minimum and Maximum pass. |
| Outside | Yes | Yes | Elements with output values less than Minimum or greater than Maximum pass. |

#### Labels

When enabled, the elements which pass the specified criteria will be labeled.

#### Abs Value

If this option is enabled, the absolute value of the output data is compared to the specified criteria.

#### Maximum and Minimum...

...allow you to dynamically increase or decrease *Maximum* and/or *Minimum* values using the appropriate slider bar.

- Added ability to transform line element output into a selected Coordinate System. A typical “use case” for this type of transformation is to view results from Nastran Spring/Damper elements (CBUSH) in a single coordinate system, even though the elements may be oriented in different coordinate systems.

- Added *"7..Visible Min/Max"* option to the *Level Modes* section for the *Contour/Criteria Levels* option in *View, Options*. This option considers ALL visibility options currently specified for the current view to automatically determine the maximum and minimum values from the *Contour* Output Vector currently displayed, then interpo-lates between them to determine all intermediate values. These visibility options include visibility check boxes for Property, Material, Element Type, and/or Element Shape; visibility of individual elements; visible Groups; visible Layers; and/or elements being drawn/erased by the *Draw/Erase* toolbar.

- Added warning when attempting to transform data in a Output Set which was created using either the RSS Combination or Envelope methods of the *Model, Output, Process* command. The warning will also be issued for an Output Set created by the RSS Combination or Envelope methods available for a *Results Set Processing Table* in the *Function/Table Editor* or *Results Set Processing Data Surface* in the *Data Surface Editor*, when the output is stored in the database.

- Updated Post Titles to show End A and End B output vectors when displaying a contour on line element.

# Geometry Interfaces

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

| FEMAP Interface | Latest Supported Version |
|---|---|
| Parasolid | 31.0 |
| Solid Edge | 2019 |
| NX | 12.0 |
| ACIS | 2018 1.0 |
| CATIA V5 | V5-6 R2018 |
| Pro/Engineer | Creo 5 |
| SolidWorks | 2018 |
| JT | 9.0 |

For details, see Section 9, “Geometry Interfaces” in the *FEMAP User Guide*.

- Updated *File, Import, Geometry* command in FEMAP 12.0.1 to automatically scale geometry to insure all geometry resides in the valid modeling region, which allows further operations to work properly. An attempt is made to update solids to the preferred modeling scale factor, provided all solids end up inside modeling region.
- Updated the *File, Export, Geometry* command by adding the *Topology Optimization* option, which requires output from a NX Nastran Topology Optimization analysis to export a STL file of the optimized shape. To dynamically specify elements to use for the faceted representation, click the *Options* button, then use the *Topology Optimization STL File Output* dialog box, which is very similar to the one used by the *View, Advanced Post, Dynamic Criteria* command. See Output and Post-Processing for more information.

# Analysis Program Interfaces

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

- FEMAP Neutral File Interface
- Analysis Set Manager
- NX Nastran Interface
- Nastran Interfaces (NX and MSC/MD)
- MSC Nastran Interface
- ANSYS Interface
- ABAQUS Interface
- 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 v12.0 changes
- Updated *File, Import, Femap Neutral* command to better handle cases where geometric entities have inconsistent geometry scale factors.

## Analysis Set Manager

- Updated *Analysis Set* dialog box, only when running in “FEMAP with NX Nastran” mode, to check for existence of a *Linked Solver* when switching *Analysis Program* from non-NX Nastran to “36..NX Nastran”. If dialog box is already set to “Linked Solver”, and a linked solver for NX Nastran is specified, nothing will change. Otherwise, it will change to *Integrated Solver*. In all other cases nothing is changed.

## NX Nastran Interface

- Added ability to Read / Write output requests for Elastic Strain (ELSTRN) and Thermal Strain (THSTRN), as well as renaming the Strain item in the NASTRAN Output Requests dialog box to Total Strain.

- Added support for SOL 401 (Multi-Step Structural Solution) that supports a combination of static (linear or nonlinear), modal (real eigenvalue), and bolt pre-load subcases. Specifically, this includes adding the *Multi-Step Control Options* dialog box, along with the *Solution and Convergence Options*, *Contact/Bolt Preload Control Options*, and *Creep Options* dialog boxes to the *Analysis Set Manager* when *Analysis Type* is set to "27..Multi-Step Structural" to write the NLCNTL entry for the Master Case and/or subcases. In addition, added *Analysis Type* drop-down to the *Master Requests and Conditions* dialog box, which is used to select the type of analysis for the Master Case, and writes the appropriate ANALYSIS = entry to Case Control.
- Added support for SOL 402 (Multi-Step Nonlinear Kinematic Solution) that supports a combination of subcase types (static linear, static nonlinear, nonlinear dynamic, bolt pre-load, modal, Fourier, buckling) and large rotation kinematics. Specifically, this includes adding the *Multi-Step Control Options* dialog box, along with the *Solution and Convergence Options* dialog box to the *Analysis Set Manager* when *Analysis Type* is set to "28..Multi-Step Nonlinear Kinematic" to write the NLCNTL2 entry for the Master Case and/or subcases. In addition, this includes adding the *Multi-Step Global Control Options* dialog box to optionally write the NLCNTLG entry to the Master Case.
- Added *Subcase (ID) Time Steps* dialog box to the *Analysis Set Manager* when *Analysis Type* is set to "27..Multi-Step Structural" (SOL 401) or "28..Multi-Step Nonlinear Kinematic" (SOL 402) to write TSTEP1 to the Master Case (*ID* = 0) or subcase.
- Added *Analysis Type* drop-down to the *Analysis Case* dialog box in the *Analysis Set Manager* when *Analysis Type* set to "27..Multi-Step Structural" (SOL 401) or "28..Multi-Step Nonlinear Kinematic" (SOL 402). This is used to specify the type of analysis for a subcase, which writes the appropriate ANALYSIS = entries to the Case Control section. In addition, added the *Step Control* section to the *Analysis Case* dialog box, which is used to select *Sequentially Dependent* or *Not Sequentially Dependent*, and writes the SEQDEP entry to the appropriate subcase in Case Control.
- Added *MATNL* and corresponding drop-down to *NASTRAN Bulk Data Options* dialog box, which is only available in the *Analysis Set Manager* when *Analysis Type* is set to "27..Multi-Step Structural" (SOL 401) or "28..Multi-Step Nonlinear Kinematic" (SOL 402). Writes PARAM,MATNL with the selected option.
- Added ability to create Connector Set for support of Multi-Step Structural (SOL401) and Multi-Step Nonlinear Kinematic (SOL402). Connector Sets can be selected in the *Contact Sets* and/or *Glue Sets* sections of the *Boundary Conditions* dialog box in *Analysis Set Manager* when *Analysis Type* is set to "27..Multi-Step Structural" (SOL 401) or "28..Multi-Step Nonlinear Kinematic" (SOL 402) for the Master Case. For subcases, only the *Contact Sets* section is available. See Connections (Regions, Properties, and Connectors) section.
- Added tabs in Connection Property dialog box for support of Multi-Step Structural (SOL401) and Multi-Step Nonlinear Kinematic (SOL402). See Connections (Regions, Properties, and Connectors) section.
- Added read/write support for Cohesive Elements (CHEXCZ, CPENTCZ), along with the Cohesive Property (PSOLCZ) and Cohesive Material (MATCZ). See Elements, Materials, and Properties sections.
- Added support for Topology Optimization in Design Optimization (SOL 200), including ability to define Topology Optimization regions (DETVEL1 and associated GROUP entries) and specify Manufacturing Constraints (DMNCON entries). See Optimization section.
- Added support for reading ply-by-ply laminate results created by random response analysis, from the .op2 file. This includes ply-by-ply Stresses including von Mises Stress for PSDF (OESPSD1C), Cumulative Root Mean Square output (OESXNO1C), and Positive Crossing (OESCRM1C) output sets, along with ply-by-ply Strains for PSDF (OSTPSD1C) and Cumulative Root Mean Square (OSTCRM1C) output sets.
- Added support for Strength Ratio output in new format generated by NX Nastran 12 for certain conditions.
- Updated the *Rigid Element Thermal Expansion* option in *Translator Options* section of *NASTRAN Bulk Data Options* dialog box in the *Analysis Set Manager* to be *Rigid Element Method* with a corresponding drop-down. When *Analysis Type* is set to "27..Multi-Step Structural" (SOL 401) or "28..Multi-Step Nonlinear Kinematic" (SOL 402), select from "0..AUTO", "2..LINEAR", or "3..STIFF". When *Analysis Type* is set to other supported analysis types, select from "1..LAGRAN" or "2..LINEAR". Writes the RIGID = entry to the Master Case.

A number of bugs were corrected

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

## *Nastran Interfaces (NX and MSC/MD)*

- Added support for Optimization Variables for Materials (DVMREL1 entries) and Elements (DVEREL1 and associated GROUP entries), along with adding support for many more for Properties (DVPREL1 entities) for Design Optimization (SOL 200). See Optimization section.
- Added support for more types of Optimization Limits (DRESP1, DCONSTR, and DCONADD entries) for Design Optimization (SOL 200). See Optimization section.
- Added check when reading Coordinate Systems, Nodes and Elements to issue error and skip the entities if the ID is greater than 99,999,999.
- Updated process when importing or attaching to .op2 files created by response spectrum analysis to always skip principal stress and total vector calculations.
- Updated process when importing or attaching to .op2 files to first sort output sets by subcase, then by time step.

## *MSC Nastran Interface*

- Added Support for output from Fastener Elements (CFAST).
- Added support for SRCOMPS parameter.
- Added support for MSC Nastran Strength Ratio output.
- Added Database read / write support for MSC BCTABLE items: HCT, HCV, HNC, BNC, EMISS. These can only be accessed via the API.
- Updated the *Rigid Element Thermal Expansion* option in *Translator Options* section of *NASTRAN Bulk Data Options* dialog box in the *Analysis Set Manager* to be *Rigid Element Method* with a corresponding drop-down. When *Analysis Type* is set to a supported analysis type, select from “1..LAGRAN”, “2..LINEAR”, or “4..LGELIM”. Writes the RIGID = entry to the Master Case.

A number of bugs were corrected

For details, see Section 8, “Analysis Program Interfaces” in the *FEMAP User Guide*.

## *ANSYS Interface*

Added an entirely new ANSYS interface for version 12, which includes support for modern element types, along with other modern inputs and options. The new translator, and all user interface components throughout FEMAP which were updated to support the new translator, are available when the *Enable Legacy Ansys Interface* option is DISABLED in the *General Solver Options* section on the *Interfaces* tab of the *File, Preferences* command.

New features include:

- Analysis Set Manager - New dialog boxes to specify options for different types of analysis in any subcase. These include the *Ansys Analysis Case*, *ANSYS Load Step*, *ANSYS Time Step Options*, *ANSYS Modal Analysis Options*, *ANSYS Buckling Analysis Options*, *ANSYS Transient Dynamics Options*, *ANSYS Harmonic Analysis Options*, *ANSYS Nonlinear Options*, *ANSYS Output Requests*. In addition, the type of analysis being performed by each “case” can be selected using the *Analysis Type* drop-down in the *Ansys Analysis Case* dialog box.
- Connection Property - ANSYS tab has been updated to offer more options than were previously available. See Connections (Regions, Properties, and Connectors).
- Elements and Properties - 40 of 42 element/property types in FEMAP are mapped to modern ANSYS elements.
- Element Formulations - An *Ansys* tab has been added to the *Element Formulation* dialog box, which is used to specify element type (*Ansys Option* section) and up to six KEYOPT values for each element type (*Ansys Keyopt* section).
- Materials - 30 Material types are mapped to modern ANSYS material models (constant or tabular values).
- Loads and Boundary Conditions - High-level of support for different boundary and loading conditions on nodes, elemental, and geometric entities (constant of tabular values)
- API - All options in new ANSYS interface available to be set using Application Programing Interface (API).

To use the “legacy interface”, enable the *Enable Legacy Ansys Interface* preference.

A number of bugs were corrected in the legacy translator.

For details, see Section 8, “Analysis Program Interfaces” in the *FEMAP User Guide*.

### ABAQUS Interface

- Added support for analytical rigid surfaces of type CYLINDER, SEGMENTS, or REVOLUTION. See Connections (Regions, Properties, and Connectors).
- Added Support For ABAQUS 2018 ODB results files.
- Added reading Pore Pressure “POR” results from the ODB results file.
- Added export of CBUSH elements as ABAQUS CONN3D2 elements, which reference *CONNECTOR BEHAVIOR and *CONNECTOR ELASTICITY entries. In addition, added *Write CBUSH as Matrix* option to *ABAQUS Model Options* dialog box, which when enabled, writes out CBUSH elements *MATRIX INPUT and *MATRIX ASSEMBLE entries.

A number of bugs were corrected.

For details, see Section 8, “Analysis Program Interfaces” in the *FEMAP User Guide*.

### DYNA Interface

- Added support for *CONTROL_TIMESTEP entry for both Explicit Transient Dynamics and Implicit Transient Dynamics. For both types of analysis, this is specified by clicking the *Advanced* button in the *Solver Options* section of the *LS-DYNA Analysis Control* dialog box in the *Analysis Set Manager*, which opens the *LS-Dyna Solver Options - *CONTROL* dialog box.

A number of bugs were corrected

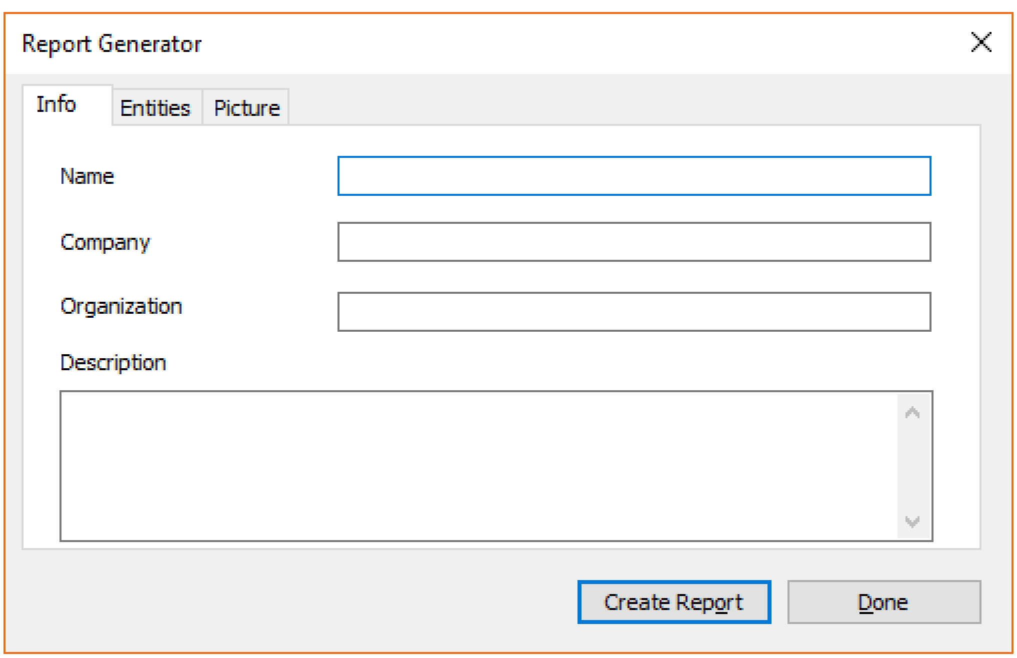
For details, see Section 8, “Analysis Program Interfaces” in the *FEMAP User Guide*.

## Tools

- Added the *Tools, Report Generator* command.

This tool can be used to automatically generate a report by automatically connecting to Microsoft Word, creating a document, then filling the report with general information supplied by the user and information for entity types selected by the user, which may presented in table format, as a picture, or both.

What is sent to the report is controlled by the *Report Generator* dialog box:



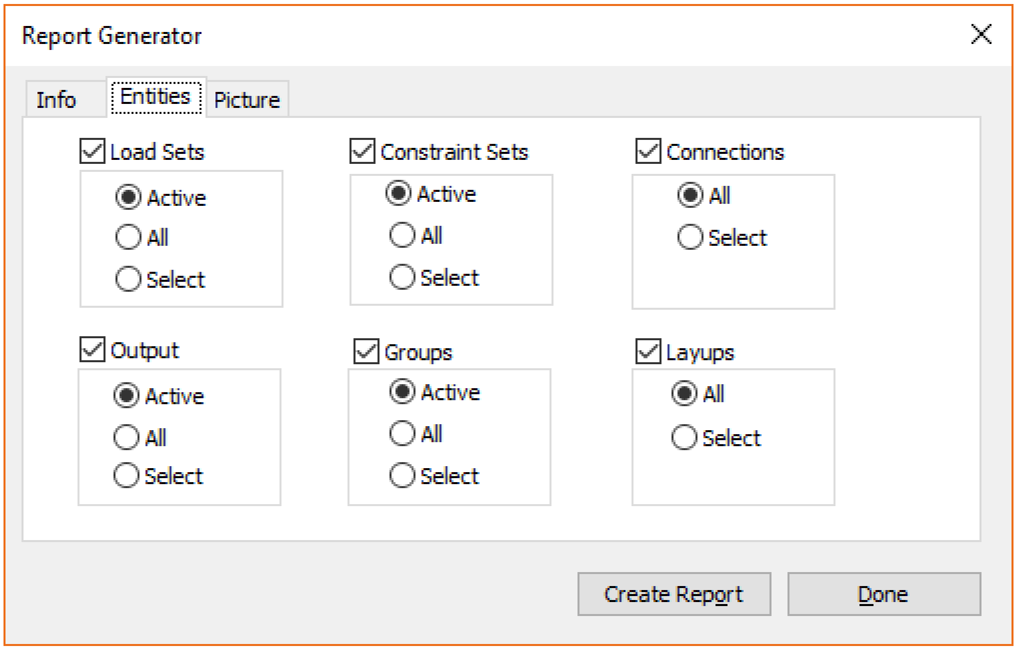

This dialog box has three tabs: *Info*, *Entities*, and *Picture*, each of which will explained further.

### Info

This tab is used to enter some general information, such as the *Name* of the person working on the model, the *Company* they work for and *Organization* they are a member of, along with a *Description* of the model.

### Entities

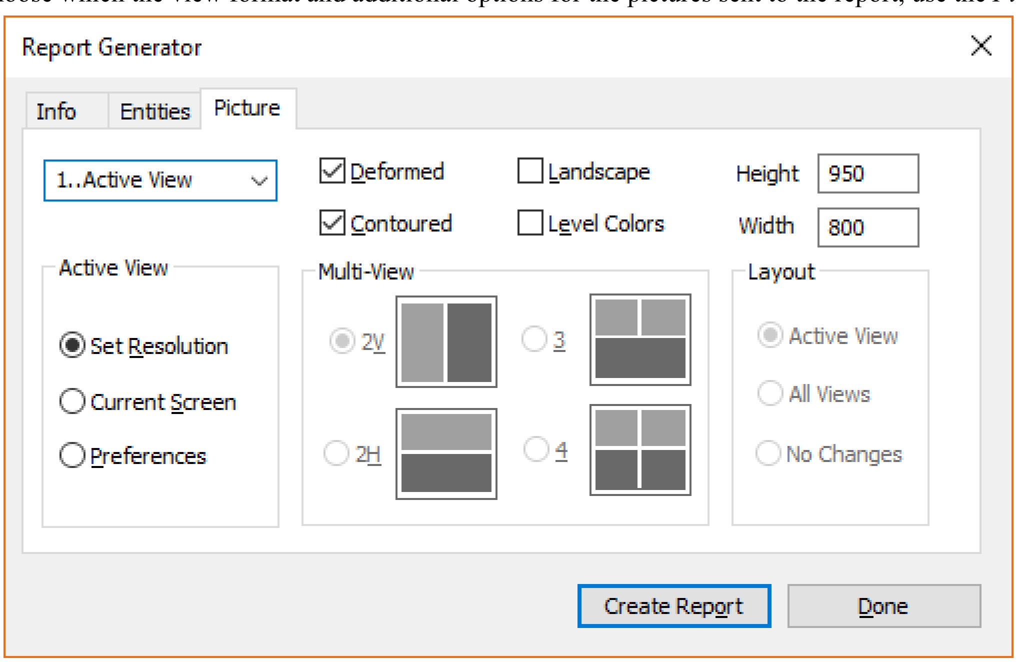
To to select which entity types to send to the report, along with some options for which of those entities to send, use the *Entities* tab:




Use the check boxes next to *Load Sets*, *Constraint Sets*, *Connections*, *Output*, *Groups*, and/or *Layups* to send information about that entity type to the report. Each entity type also offers the ability to send *All* entities of that type or *Select* which entities of each type to send, using an appropriate selection dialog box. In addition, it may make sense to only send information about the *Active* entity of specific type, so that is also an option for *Load Sets*, *Constraint Sets*, *Output*, and *Groups*.

### Picture

To choose which the view format and additional options for the pictures sent to the report, use the *Picture* tab:



Use the drop-down in the upper left of the Picture tab to choose “1..Active View”, “2..Multi-View”, or “3..Current Layout”, then select options in the corresponding section.

For “1..Active View”, choose *Set Resolution* in the *Active View* section to use the *Height* and *Width* specified in this dialog box, *Current Screen* to use the height and width of the current graphics window, or *Preferences* to use the options specified in the *Picture Save Defaults* section of the *Views* tab found in the *Preferences* dialog box. For more information, see Section 2.6.2.2, "Views".

For “2..Multi-View”, choose *2V* (2 Vertical), *2H* (2 Horizontal), *3*, or *4* in the *Multi-View* section to create that many copies of the “active view” in the orientations shown in the dialog box.

For “3..Current Layout”, choose send the only the *Active View*, *All Views* in the model, or the current layout with *No Changes*, which may be showing multiple views in any orientation to the report.

When the Landscape option is enabled, all pictures will be transferred using Landscape orientation, while they will be transferred in Portrait mode when Landscape is disabled.

Finally, the *Deformed*, *Contoured*, and *Level Colors* options are only be available when sending output information. to the report, and control if the picture will show the model deformed (enabled) or not deformed (disabled) and/or contoured (enabled) or with no contours (disabled) using either Level Colors (enabled) or Continuous Colors (disabled).

## *Model Merge*

- Added *Copy in Current Model* option to the *Merge/Extract* section. This option allows entities from a model to be merged into the same model. Typically, this would be used in conjunction with one of the *Orientation/Transform* options to create duplicate entities in a different location.
- Added *Keep Loads and Constraints in Original Sets* option to *Options* section. When enabled, which is the default when using the *Copy in Current Model* option, load sets and constraint sets will not be renumbered. Instead, any load set or constraint set which exists in both the *From Model* and *To Model* will be combined into a single load set or single constraint set in the *To Model*, using the original ID.
- Added *Orientation/Transform* section, which replaces the *Transform Merged Model* option and corresponding drop-downs to select coordinate systems.

This section is used to orient the entities currently in the *Entities to Merge* list in the active or current model.

***Note:*** When using *Transform Merged Model*, the *Output Set* entity type must NOT be selected. If selected, the command will issue “Unable to transfer results when transforming a model during merge.” message.

***None*** - specifies the merged/copied entities will not be reoriented and/or transformed.

***Move Along Vector*** - specifies the merged/copied entities will be transformed along a vector specified in the active or current model using the standard vector definition dialog box. See Section 3.5.1, "Geometry, Copy Commands" or Section 5.4.1, “Mesh, Copy Menu” for more information.

***Rotate Around Vector*** - specifies the merged/copied entities will be rotated around a vector specified in the active or current model using the standard vector definition dialog box. After specifying the rotation vector, enter a *Rotation Angle* and optionally a *Translation Distance* in the *Rotation and Translation* dialog box. See Section 3.5.4, "Geometry, Rotate Commands" or Section 5.4.4, "Mesh, Rotate Menu" for more information.

***Reflect Across Plane*** - specifies the merged/copied entities will be reflected across a plane specified in the active or current model using the standard plane definition dialog box. See Section 3.5.5, "Geometry, Reflect Commands" or Section 5.4.5, "Mesh, Reflect Menu" for more information.

***Between Vectors*** - specifies the merged/copied entities will be “aligned” using two vectors, each specified using a standard vector definition dialog box. The first vector, which is specified in the *From Model*, defines the original position and orientation that will be aligned. The second vector, which is specified in the *To Model*, defines the new or desired position and orientation. When using *Copy in Current Model*, both vectors are defined in the same model. The entities are first translated from the origin of the first vector to the origin of the second vector, then rotated based on the angle between the vectors.

***Between Planes*** - specifies the merged/copied entities will be “aligned” using two planes, each specified using a standard plane definition dialog box, one in the *From Model*, one in the *To Model*. When using *Copy in Current*

*Model*, both planes are defined in the same model. The behavior is similar to *Between Coordinate Systems*, only each specified plane (X and Y axes) and each plane's normal direction (Z axis) are used to determine XYZ axes to align.

***Between Coordinate Systems*** - specifies the merged/copied entities will be "aligned" using two coordinate systems selected using the *From* and *To* drop-downs. The XYZ axes of the *From* coordinate system, as it is defined in the *From Model*, will be aligned to the XYZ axes of the *To* coordinate system, as it is defined in the *To Model*. All the selected and associated entities will move as a rigid body to the newly aligned location. When using *Copy in Current Model*, both coordinate systems are selected from the same model. The type of coordinate system used for *From* and *To* does not matter, as only the axes are aligned.

***Note:*** If using a cylindrical coordinate system, the R (radial) axis is the X axis, the T (theta) axis corresponds is the Y axis, and the Z axis is the Z axis. If using a spherical coordinate system, the R (radial) axis is the X axis, the P (phi) axis is the Y axis, and the T (theta) axis is the Z axis.

- Updated *File, Merge* command to skip rebuilding solid faceting of previously existing solids.
- Updated *File, Merge* to better handle cases where geometry entities have inconsistent geometry scale factors.

# OLE/COM API

### New and modified API Objects and Attributes

- Added ElementNoResultMode attribute to the View Object for FEMAP 12.0.1
- Added CopyTool (feCopyTool) object to the API. Also, added Repetitions, CopyInSameLocation, AlwaysCreateParentCSys, UsePattern, and ReflectionTrapWidth attributes to the feCopyTool Object.
- Added MoveTool (feMoveTool) object to the API. Also, added AlwaysCreateParentCSys and ReflectionTrapWidth attributes to the feMoveTool Object.
- Added Discrete Value Set (feDiscreteValueSet) object to the API. Also, added IntegerType and RangeType attributes to the feDiscreteValueSet Object.
- Added Mesh Point (feMeshPt) object to the API. Also, added DefinitionID, PointID, layer, color, OnGeometryType, OnGeometryID, NodeOnGeometry, DistanceToGeometry, Locked, LocationOnGeometry, and vLocationOnGeometry to feMeshPtObject.
- Added Mesh Point Definition (feMeshPointDefinition) object to the API. Also, added title attributes to the feMeshPointDefinition Object.
- Added Optimization Manufacturing Constraints (feOptMC) object to the API. Also, added title, ManConType, Vec1, vVec1, Vec2, vVec2, Vec3, vVec3, Vec4, vVec4, nField1, dField1, and dField2 attributes to the feOptMC Object.
- Added Optimization Relationship (Variable) (feOptRel) object to the API. Also, added title, TypeField, NameField, RelationshipType, TopologyActive, Bounds, vBounds, RelationshipID, and BoundType attributes to the feOptRel Object.
- Added Optimization Response (Limits) (feOptResp) object to the API. Also, added title, rtype, ptype, atta, attb, ResponseCategory, VectorID, attbFieldType, attiDataType, and attaBlank attributes to the feOptRel Object.
- Added SubcaseAnalysisType attribute to feAnalysisCase Object.
- Added NasModeOn, NasModeEigrSet, NasModeMethod, NasModeSolutionType, NasModeEstRoots, NasModeDesiredRoots, NasModeNormOpt, NasModeNormNode, NasModeNormDOF, NasModeMassForm, NasModeXYOn, NasModeXYRefNode, NasModeFreqRange, vNasModeFreqRange, NasModesSkipEigr, NasModeImagFreqRange, vNasModeImagFreqRange, NasModesComplexConverge, NasModesComplexRegionWidth, NasModeDampOverall, NasModeXYRequest, and vNasModeXYRequest attributes for Nastran Modal Analysis subcases to feAnalysisCase Object.
- Added NasMsnlCntOn, NasMsnlCntSkipNLCNTL, vNasMsnlCntConv_flags, vNasMsnlCntConv_value, NasMsnlCntMaxbis, NasMsnlCntMaxdiv,NasMsnlCntMaxiter, NasMsnlCntMaxqn, NasMsnlCntEpsbolt, NasMsnlCntZerbolt, NasMsnlCntItrbolt, NasMsnlCntMisfblt, NasMsnlCntMsglvlb, NasMsnlCntLvar, NasMsnlCntMsglvl, NasMsnlCntSolver, NasMsnlCntThrmst, NasMsnlCntTvar, NasMsnlCntFollowk, NasMsnlCntKupdate, NasMsnlCntSpink, NasMsnlCntStfoptn, NasMsnlCntStressk, NasMsnlCntTstepk,

NasMsnlCntCntmdiv, NasMsnlCntFsymtol, NasMsnlCntKsym, NasMsnlCntKsymtol, NasMsnlCntMsglvlc, NasMsnlCntUsolver, NasMsnlCntCrcerat, NasMsnlCntCrcinc, NasMsnlCntCreep, NasMsnlCntCricoff, NasMsnlCntCrinfac, NasMsnlCntCrmfmn, NasMsnlCntCrmfmx, NasMsnlCntCrteabs, NasMsnlCntdCrteco, NasMsnlCntCrterel, NasMsnlCntPlastic, NasMsnlCntAutotim, NasMsnlCntDtinit, NasMsnlCntDtmax, NasMsnlCntDtmin, NasMsnlCntDtsbcdt, NasMsnlCntEqmfmin, NasMsnlCntEqmfmx, NasMsnlCntTsccr, NasMsnlCntTsceq, NasMsnlCntTscumat, NasMsnlCntUmfmin, and NasMsnlCntUmfmx attributes for NX Nastran SOL 401 subcases to the feAnalysisCase Object.

- Added NasMsnlkCnt2on, NasMsnlkCnt2SkipNLCNTL2, NasMsnlkCnt2DISLIM, NasMsnlkCnt2ROTLIM, NasMsnlkCnt2DEFLIM, NasMsnlkCnt2LVAR, NasMsnlkCnt2TVAR, NasMsnlkCnt2DIPR, NasMsnlkCnt2NORM, NasMsnlkCnt2CREEP,NasMsnlkCnt2PLASTIC, NasMsnlkCnt2STAB, NasMsnlkCnt2MADI, NasMsnlkCnt2ERCD, NasMsnlkCnt2PRED, NasMsnlkCnt2IMPL, NasMsnlkCnt2BETA, NasMsnlkCnt2GAMA, NasMsnlkCnt2ALFA, NasMsnlkCnt2TETA, NasMsnlkCnt2PRCO, NasMsnlkCnt2HPRCO, NasMsnlkCnt2ILNS, NasMsnlkCnt2PRLN, NasMsnlkCnt2AMIN, NasMsnlkCnt2AMAX, NasMsnlkCnt2ITMA, NasMsnlkCnt2PRCR, NasMsnlkCnt2REFP, NasMsnlkCnt2PRCQ, NasMsnlkCnt2REFU, NasMsnlkCnt2PRCE, NasMsnlkCnt2REFE, NasMsnlkCnt2IT1K, NasMsnlkCnt2IT2K, NasMsnlkCnt2IT3K, NasMsnlkCnt2PLAS, NasMsnlkCnt2CIBL, NasMsnlkCnt2DTI0, NasMsnlkCnt2HMIN, NasMsnlkCnt2HMAX, NasMsnlkCnt2RUP, NasMsnlkCnt2RDOW, NasMsnlkCnt2RSUB, NasMsnlkCnt2RELC, NasMsnlkCnt2DCON, NasMsnlkCnt2PRCS, NasMsnlkCnt2IMPG, NasMsnlkCnt2IMPR, and NasMsnlkCnt2IMPV attributes for NX Nastran SOL 402 subcases to the feAnalysisCase Object.
- Added NasMsnlTSOn, NasMsnlTSSkipTSTEP, NasMsnlTSNumIncrements, NasMsnlTSEndTime, NasMsnlTSSkipFactor, and NasMsnlTSOutputFreq attributes for NX Nastran SOL 401 and SOL 402 subcases to the feAnalysisCase Object.
- Added AnsLdstepOn, AnsKbc, AnsPstres, AnsEqslv, AnsNewConstraint, AnsNewLoad,AnsNewContact,AnsTimestepOn, AnsTime, AnsAutots, AnsNsubs, AnsNsbstep, AnsNsbmx, AnsNsbmn, AnsDtime, AnsDtmin, AnsDtmax, AnsPlslimit, AnsPlslimitvalue, AnsCrplimit, AnsCrplimitvalue, AnsDpplimit, AnsDpplimitvalue, AnsDsplimit, AnsDsplimitvalue, AnsBisect, AnsBisectfactor, AnsPredictor,AnsMidtol, AnsTolerb, AnsResfq, AnsOutputOn, AnsOutresAll,AnsOutprAll, AnsOutresAllFreq,AnsOutprAllFreq, AnsOutresAllNth, AnsOutprAllNth, AnsOutresBasic, AnsOutprBasic, AnsOutresBasicFreq, AnsOutprBasicFreq, AnsOutresBasicNth, AnsOutprBasicNth, AnsOutresRsol, AnsOutprRsol, AnsOutresRsolFreq, AnsOutprRsolFreq, AnsOutresRsolNth, AnsOutprRsolNth, AnsOutresNsol, AnsOutprNsol, AnsOutresNsolFreq, AnsOutprNsolFreq,AnsOutresNsolNth,AnsOutprNsolNth, AnsOutresVel,AnsOutprVel, AnsOutresVelFreq,AnsOutprVelFreq, AnsOutresVelNth, AnsOutprVelNth,AnsOutresAcc, AnsOutprAcc, AnsOutresAccFreq, AnsOutprAccFreq, AnsOutresAccNth, AnsOutprAccNth, AnsOutresEsol, AnsOutprEsol, AnsOutresEsolFreq, AnsOutprEsolFreq, AnsOutresEsolNth, AnsOutprEsolNth, AnsModalOptOn, AnsModalMethod, AnsModalNmode, AnsModalFreqb, AnsModalFreqe, AnsModalNrmkey,AnsModalStrmck,AnsModalRangefact, AnsModalBlocksize, AnsModalRobustlev, AnsModalCompute, AnsModalReusekey, AnsModalSymmeth, AnsBuckleOptOn, AnsBuckleMethod,AnsBuckleNmode, AnsBuckleShift, AnsBuckleLdmulte,AnsBuckleRangekey,AnsBuckleStrmck, AnsHarmonicOptOn, AnsHarmonicMethod, AnsHarmonicMinmode, AnsHarmonicMaxmode, AnsHarmonicFreqb, AnsHarmonicFreqe, AnsHarmonicLogopt,AnsHarmonicReimky,AnsHarmonicClust, AnsTransOptOn, AnsTransMethod, AnsTransMinmode, AnsTransMaxmode, AnsTransTintopt, AnsTransDmpsfreq, AnsTransLumpm, AnsTransAlphad, AnsTransBetad, AnsTransDmpstr, AnsTransDmprat, AnsTransGamma, AnsTransGammavalue, AnsTransAlpha, AnsTransDelta, AnsTransAlphaf, AnsTransAlpham, AnsNlOptOn, AnsNropt, AnsNeqit, AnsCnvtolU, AnsUtoler, AnsCnvtolRot, AnsRottoler, AnsCnvtolF, AnsFtoler, AnsCnvtolM, AnsMtoler, AnsCnvtolDvol, AnsDvoltoler, AnsCnvtolHdsp, AnsHdsptoler, AnsNlgeom, AnsArclen, AnsLnsrch, and AnsPred attributes for ANSYS cases to the feAnalysisCase Object.
- Added AbaModCbushAsMatrix attribute for ABAQUS Master Case to the feAnalysisMgr Object.
- Added NasMsnlCntOn, NasMsnlCntSkipNLCNTL, vNasMsnlCntConv_flags, vNasMsnlCntConv_value, NasMsnlCntMaxbis, NasMsnlCntMaxdiv,NasMsnlCntMaxiter, NasMsnlCntMaxqn, NasMsnlCntEpsbolt, NasMsnlCntZerbolt, NasMsnlCntItrbolt, NasMsnlCntMisfblt, NasMsnlCntMsglvlb, NasMsnlCntLvar, NasMsnlCntMsglvl, NasMsnlCntSolver, NasMsnlCntThrmst, NasMsnlCntTvar, NasMsnlCntFollowk, NasMsnlCntKupdate, NasMsnlCntSpink, NasMsnlCntStfoptn, NasMsnlCntStressk, NasMsnlCntTstepk, NasMsnlCntCntmdiv, NasMsnlCntFsymtol, NasMsnlCntKsym, NasMsnlCntKsymtol, NasMsnlCntMsglvlc, NasMsnlCntUsolver, NasMsnlCntCrcerat, NasMsnlCntCrcinc, NasMsnlCntCreep, NasMsnlCntCricoff, NasMsnlCntCrinfac, NasMsnlCntCrmfmn, NasMsnlCntCrmfmx, NasMsnlCntCrteabs, NasMsnlCntdCrteco, NasMsnlCntCrterel, NasMsnlCntPlastic, NasMsnlCntAutotim, NasMsnlCntDtinit, NasMsnlCntDtmax,

NasMsnlCntDtmin, NasMsnlCntDtsbcdt, NasMsnlCntEqmfmin, NasMsnlCntEqmfmx, NasMsnlCntTsccr, NasMsnlCntTsceq, NasMsnlCntTscumat, NasMsnlCntUmfmin, and NasMsnlCntUmfmx attributes for NX Nastran SOL 401 Master Case to the feAnalysisMgr Object.

- Added NasMsNLKGlobalOn, NasMsNLKGlobalRESO, NasMsNLKGlobalSTRMEAS, NasMsNLKGlobalIREF, NasMsNLKGlobalINLY NasMsnlkCnt2on, NasMsnlkCnt2SkipNLCNTL2, NasMsnlkCnt2DISLIM, NasMsnlkCnt2ROTLIM, NasMsnlkCnt2DEFLIM, NasMsnlkCnt2LVAR, NasMsnlkCnt2TVAR, NasMsnlkCnt2DIPR, NasMsnlkCnt2NORM, NasMsnlkCnt2CREEP,NasMsnlkCnt2PLASTIC, NasMsnlkCnt2STAB, NasMsnlkCnt2MADI, NasMsnlkCnt2ERCD, NasMsnlkCnt2PRED, NasMsnlkCnt2IMPL, NasMsnlkCnt2BETA, NasMsnlkCnt2GAMA, NasMsnlkCnt2ALFA, NasMsnlkCnt2TETA, NasMsnlkCnt2PRCO, NasMsnlkCnt2HPRCO, NasMsnlkCnt2ILNS, NasMsnlkCnt2PRLN, NasMsnlkCnt2AMIN, NasMsnlkCnt2AMAX, NasMsnlkCnt2ITMA, NasMsnlkCnt2PRCR, NasMsnlkCnt2REFP, NasMsnlkCnt2PRCQ, NasMsnlkCnt2REFU, NasMsnlkCnt2PRCE, NasMsnlkCnt2REFE, NasMsnlkCnt2IT1K, NasMsnlkCnt2IT2K, NasMsnlkCnt2IT3K, NasMsnlkCnt2PLAS, NasMsnlkCnt2CIBL, NasMsnlkCnt2DTI0, NasMsnlkCnt2HMIN, NasMsnlkCnt2HMAX, NasMsnlkCnt2RUP, NasMsnlkCnt2RDOW, NasMsnlkCnt2RSUB, NasMsnlkCnt2RELC, NasMsnlkCnt2DCON, NasMsnlkCnt2PRCS, NasMsnlkCnt2IMPG, NasMsnlkCnt2IMPR, and NasMsnlkCnt2IMPV attributes for NX Nastran SOL 402 Master Case to the feAnalysisMgr Object.
- Added NasMsnlTSOn, NasMsnlTSSkipTSTEP, NasMsnlTSNumIncrements, NasMsnlTSEndTime, NasMsnlTSSkipFactor, and NasMsnlTSOutputFreq attributes for NX Nastran SOL 401 and SOL 402 Master Case to the feAnalysisMgr Object.
- Added NasOptimCycles,NasOptimOn, NasOptimIsTopology, NasBulkMatnl, NasOptimMinDesobj, NasOptimDresp, NasOptimGoal, NasOptimIntervalVal, NasOptimEcho, NasOptimInterval, NasOptimMCDelayVal, NasOptimMCDelay, and NasOptimIntervalVal attributes for NX Nastran and MSC Nastran SOL 200 Master Case to the feAnalysisMgr Object.
- Added AnsLdstepOn, AnsKbc, AnsPstres, AnsEqslv, AnsNewConstraint, AnsNewLoad,AnsNewContact,AnsTimestepOn, AnsTime, AnsAutots, AnsNsubs, AnsNsbstep, AnsNsbmx, AnsNsbmn, AnsDtime, AnsDtmin, AnsDtmax, AnsPlslimit, AnsPlslimitvalue, AnsCrplimit, AnsCrplimitvalue, AnsDpplimit, AnsDpplimitvalue, AnsDsplimit, AnsDsplimitvalue, AnsBisect, AnsBisectfactor, AnsPredictor,AnsMidtol, AnsTolerb, AnsResfq, AnsOutputOn, AnsOutresAll,AnsOutprAll, AnsOutresAllFreq,AnsOutprAllFreq, AnsOutresAllNth, AnsOutprAllNth, AnsOutresBasic, AnsOutprBasic, AnsOutresBasicFreq, AnsOutprBasicFreq, AnsOutresBasicNth, AnsOutprBasicNth, AnsOutresRsol, AnsOutprRsol, AnsOutresRsolFreq, AnsOutprRsolFreq, AnsOutresRsolNth, AnsOutprRsolNth, AnsOutresNsol, AnsOutprNsol, AnsOutresNsolFreq, AnsOutprNsolFreq,AnsOutresNsolNth,AnsOutprNsolNth, AnsOutresVel,AnsOutprVel, AnsOutresVelFreq,AnsOutprVelFreq, AnsOutresVelNth, AnsOutprVelNth,AnsOutresAcc, AnsOutprAcc, AnsOutresAccFreq, AnsOutprAccFreq, AnsOutresAccNth, AnsOutprAccNth, AnsOutresEsol, AnsOutprEsol, AnsOutresEsolFreq, AnsOutprEsolFreq, AnsOutresEsolNth, AnsOutprEsolNth, AnsModalOptOn, AnsModalMethod, AnsModalNmode, AnsModalFreqb, AnsModalFreqe, AnsModalNrmkey,AnsModalStrmck,AnsModalRangefact, AnsModalBlocksize, AnsModalRobustlev, AnsModalCompute, AnsModalReusekey, AnsModalSymmeth, AnsBuckleOptOn, AnsBuckleMethod,AnsBuckleNmode, AnsBuckleShift, AnsBuckleLdmulte,AnsBuckleRangekey,AnsBuckleStrmck, AnsHarmonicOptOn, AnsHarmonicMethod, AnsHarmonicMinmode, AnsHarmonicMaxmode, AnsHarmonicFreqb, AnsHarmonicFreqe, AnsHarmonicLogopt,AnsHarmonicReimky,AnsHarmonicClust, AnsTransOptOn, AnsTransMethod, AnsTransMinmode, AnsTransMaxmode, AnsTransTintopt, AnsTransDmpsfreq, AnsTransLumpm, AnsTransAlphad, AnsTransBetad, AnsTransDmpstr, AnsTransDmprat, AnsTransGamma, AnsTransGammavalue, AnsTransAlpha, AnsTransDelta, AnsTransAlphaf, AnsTransAlpham, AnsNlOptOn, AnsNropt, AnsNeqit, AnsCnvtolU, AnsUtoler, AnsCnvtolRot, AnsRottoler, AnsCnvtolF, AnsFtoler, AnsCnvtolM, AnsMtoler, AnsCnvtolDvol, AnsDvoltoler, AnsCnvtolHdsp, AnsHdsptoler, AnsNlgeom, AnsArclen, AnsLnsrch, and AnsPred attributes for ANSYS to the feAnalysisMgr Object.
- Added CombinationType and AbsoluteCombination attributes to feChart Object.
- Added IsCombination attribute to feConnect Object.
- Added attrTopology, MeshPointID, and vMeshPointID attributes to feCurve Object.
- Added InPlanePropExcMemBend, vInPlanePropExcMemBend, BendingPropExcMemBend, and vBendingPropExcMemBend attributes to feLayup Object.
- Added UseCriteria attribute to feMapOutput Object.

- Added attrTopology, attrMidsideOnGeometry and attrMaxMidsideAngle attributes to feSolid Object.
- Added FeatureLineAngle, ViewLegendJustification, PostTitlesJustification, ContourLegendHorizontal, ContourLegendReversed, ViewLegendLocation, vViewLegendLocation, PostTitlesLocation, vPostTitlesLocation, ContourLegendLocation, vContourLegendLocation, ContourLegendWidthPct, DiscreteValueOn, DiscreteValueSetID, ModelDataContourOn, ModelDataContourGroup, ModelDataContourLabels, ViewLegendLockJustification, PostTitlesLockJustification, ContourLegendTopLeftLabel, and ContourLegendLockOrientation, attributes to the to feView Object. Also, updated AxisLocation and vAxisLocation.
- Updated label attribute on the feAeroSurf Object.
- Updated formulation and vformulation attributes on the feElem Object.

### New and Updated API Methods

- Added GetOptLimits and PutOptLimits for NX Nastran and MSC Nastran SOL 200 subcases to feAnalysisCase Object.
- Added KBC, PSTRES, EQSLV, TIME, AUTOTS, NSUBST, DELTIM, CUTCONTROL, MIDTOL, OUTRES, OUTPR, MODOPT, LANBOPTION, PCGOPT, SUBOPT, SNOPTION, QRDOPT, BUCOPT, HROPT, HARFRQ, HROUT, TRNOUT, LUMPM, ALPHAD, BETAD, DMPSTR, DMPRAT, TINTP, NROPT, NEQIT, CNVTOL, NLGEOM, ARCLEN, LNSRCH, and PRED for ANSYS Cases to feAnalysisCase Object.
- Added GetOptVars, PutOptVars, GetOptLimits, PutOptLimits, GetOptMCs, and PutOptMCs for NX Nastran and MSC Nastran SOL 200 Master Case to feAnalysisMgr Object.
- Added KBC, PSTRES, EQSLV, TIME, AUTOTS, NSUBST, DELTIM, CUTCONTROL, MIDTOL, OUTRES, OUTPR, MODOPT, LANBOPTION, PCGOPT, SUBOPT, SNOPTION, QRDOPT, BUCOPT, HROPT, HARFRQ, HROUT, TRNOUT, LUMPM, ALPHAD, BETAD, DMPSTR, DMPRAT, TINTP, NROPT, NEQIT, CNVTOL, NLGEOM, ARCLEN, LNSRCH, and PRED for ANSYS Master Case to feAnalysisMgr Object.
- Added ConnectionsByType, CombinationsByType, IsGlueConnector, and GetCombinations to feConnect Object.
- Added SetFriction, GetFriction, SetAnalyticSurfaceType, and GetAnalyticSurfaceType to feContact Object.
- Added Option Methods (Clear, SetNumbering, SetVectorPattern, SetCSysPattern, SetPropertyOption, IncludeOtherEntities, and AllOtherEntities) and Operation Methods (AlongVector, PointToPoint, BetweenCSys, BetweenVectors, BetweenPlanes, RotateAroundVector, RotatePointToPoint, Reflect, CreatePattern, and InPlace) to feCopyTool Object.
- Added MeshPointID, vMeshPointID, and InitMeshAttributes to feCurve Object.
- Added GetDataArray to feDataSurf Object.
- Added Get, Put, GetNumberLevels, GetLevelInteger, SetLevelInteger, AddLevelInteger, GetLevelFloat, SetLevelFloat, AddLevelFloat, Sort, GetTitle, SetTitle, CreateRangeInteger, CreateRangeFloat, and AutoCreateLevels to the feDiscreteValueSet object.
- Added GetAnsysFormulationArray, PutAnsysFormulationArray, GetAnsysKeyoptArray, PutAnsysKeyoptArray, HasProp, and HasMatl to feElem Object.
- Added SetTotalLoad to feLoadDef Object.
- Added Put, Enable, IsEnabled, OnPoint, AutoDistanceToGeometry, UpdateMeshPoints, and UpdateMeshPointsOnGeometry to feMeshPt object.
- Added Get, Delete, ResetNextMeshPoint, NextMeshPoint, and CountMeshPoints to feMeshPointDefinition object.
- Added Option Methods (Clear and IncludeOtherEntities) and Operation Methods (AlongVector, PointToPoint, BetweenCSys, BetweenVectors, BetweenPlanes, RotateAroundVector, RotatePointToPoint, Reflect) to feMoveTool Object.
- Added SetAddm, GetAddm, SetCast, GetCast, SetCheckerboard, GetCheckerboard, SetCycSym, GetCycSym, SetExtrusion, GetExtrusion, SetMinSize, GetMinSize,SetMaxSize, GetMaxSize, SetPlSym, and GetPlSym to feOptMC object.

- Added SetupTopology, GetTopology, SetupShellT, GetShellT, SetupProp, GetProp, SetupMatl, GetMatl, and CalculateBounds to feOptRel object.
- Added Get, Put, SetEntities, GetEntities, SetupDisplacement, GetDisplacement, SetupSPCForce, GetSPC-Force, SetupElem, GetElem, SetupESE, GetESE, SetupFreq, GetFreq, SetupEign, GetEign, SetupBuckle, Get-Buckle, SetConstraints,and, and GetConstraints to feOptResp object.
- Added HasMatl to the feProp Object.
- Added SetLineTransform to the feResults Object.
- Added AddAllInRangeInSet and AddBySize to the feSet Object.
- Added InitMeshAttributes and Volume to the feSolid Object.
- Added InitMeshAttributes to the feSurface Object.
- Added FitVisible, SetModelDataContourForm, GetModelDataContourForm, SetModelDataContourType, Get-ModelDataContourType to the feView Object.
- Added GetFirstPathID, GetNextPathID to the feViewOrient Object.
- Updated SetDefaults on the feConnectionProp Object.

#### The following functions have been added:

- feFileWriteTopologyStl
- feSolidThicken
- feSurfaceMidAuto3
- feSurfaceOffset2
- feImprintLocationInSurface
- feResultsToDataTable2
- feMeshEdgeMembers2
- feMeshPoint2
- feMeshCurve2
- feModifyLineElementDirection
- feGroupFromCriteria
- feGetElementAdjacentFaces
- feSelectOutput2
- feSelectOutputSets2
- feOutputValueGetRGB
- feAppResetLocale
- feGetNodalAreaOrLength
- feGetNodalAreaOnSurface

#### New and removed Global Variables

- Added Pref_RenderBestAvailable, Pref_NXNastEntLicenseServer, Pref_FastHoleMeshing, and Pref_EnableL-egacyAnsysInterface to set various preferences.
- Removed Pref_NastranUseILP64, as this preference is no longer available.

#### New and Updated API Scripts (Custom Tools Menu)

Added *Custom Tools, Views, View Themes* command (View Themes.exe), which runs as a dockable pane to provide easier creation and saving of backgrounds and entity colors.

Added *Custom Tools, Examples, PowerPoint - Group PostProcess Capture* command (PowerPoint - Group Post-Process Capture.bas) that automatically creates a PowerPoint presentation, then create a slide containing a picture of each selected group. The script can also connect to an existing PowerPoint presentation and start creating new slides at a specific slide number or simply create imagines and save them to the specified directory (i.e., potentially not interact with PowerPoint at all).

Updated existing *Custom Tools, Data Table, Add Corner Thickness* script. The script now works with the FEMAP Selector.

Updated existing *Custom Tools, Example, Using Text* script. Added auto-repeat if turned on in FEMAP Preferences.

Updated existing *Custom Tools, PostProcessing, GROUP_POST_FNO* script with option to export Full Model or individual Group(s).

Updated existing *Custom Tools, PostProcessing, List Output to Excel* script to use Results Browsing Object for faster Output Vector listing.

Updated existing *Custom Tools, PostProcessing, Calculate Factor of Safety* script for Solid Elements to prompt user for Limit Stress values if none are specified in material card.

Updated existing *Custom Tools, PostProcessing, Calculate Mid Plate Stresses_Results Browsing Object* script to create additional vector for Mid-Plate Von Mises Stress.

## Preferences

### Graphics

- Added *Best Possible* option to the *Graphics Options* section. This option, when enabled, attempts to use information from the computer running FEMAP (installed RAM, Graphics Card, version of Open GL, etc) to determine what should be the “best possible” settings for several items in the *Graphics Options* section. When enabled, the *Performance Graphics*, *Max VBO MB*, *Min VBO B*, and *Memory Optimization* options, along with the setting for the *Vertex Arrays drop-down*, cannot be specified by the user. When disabled, these items can specified individually

***Note:*** To work properly, this option relies on each computer to provide accurate information about a number of different systems and pieces of hardware. If any of the queried information being relayed to FEMAP is inaccurate, this may cause performance issues, thus it might be necessary to disable this option.

- Updated *Mesh Size* in the *Include In Dynamic Rotation* section to be *Mesh Locations*, as both Mesh Size on curves and Mesh Points are now not included in dynamic rotation unless this option is enabled.

### Geometry/Model

- Added *Surface Meshing in Memory* to *Meshing and Properties* section. This preference determines whether additional memory will be allocated by the FEMAP boundary mesher. If this option is selected, FEMAP will allocate new memory to create the mesh. If it is not selected, FEMAP will utilize the memory allocated in the database to perform the mesh. By allocating new memory, the FEMAP mesher can run significantly faster than if it is limited to the database memory. Therefore, this option should almost always be turned on. The only reason to turn this option off is if the available memory on the current machine is low enough that allocation of new memory is extremely limited.

### Interfaces

- Added *Enable Legacy Ansys Interface* option to the *General Solver Options* section. When this option enabled, the *Analysis Set Manager* and other dialog boxes will use the “legacy” version of the ANSYS analysis interface and translator (i.e., essentially what was in FEMAP versions prior to version 12).
- Added *Enterprise License Server* option to the *Nastran Options* section, which can be used to specify a license server if using an enterprise version of NX Nastran. Simply enter the “@” symbol followed by the name of the license server (i.e., @Nastran_Server_Name).