What's New in FEMAP 11.3

FEMAP 11.3 include enhancements and new features, which are detailed below:

User Interface

Geometry

Meshing

Elements

Properties

Loads and Constraints

Renumbering

Listing

Groups and Layers

Views

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

Tools

OLE/COM API

Preferences

FEMAP 11.2.1 and 11.2.2 include enhancements and new features, which are detailed below:

User Interface

Meshing

Aeroelasticity

Connections (Regions, Properties, and Connectors)

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

OLE/COM API

FEMAP 11.2 includes enhancements and new features, which are detailed below:

User Interface

Geometry

Meshing

Elements

Element Update

Properties

Aeroelasticity

Loads and Constraints

Connections (Region, Properties, and Connectors)

Groups and Layers

Views

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

Tools

Model Merge

OLE/COM API

Preferences

What’s New for version 11.3

User Interface

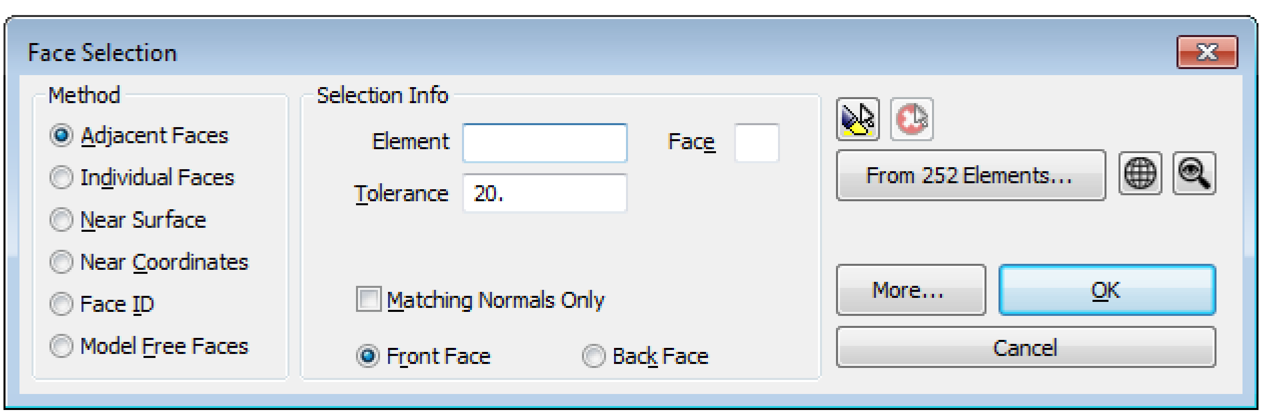
General, Menu, Toolbars, Model Info tree, Meshing Toolbox, PostProcessing Toolbox, Charting pane, Entity Editor, Connection Editor, Data Table

General

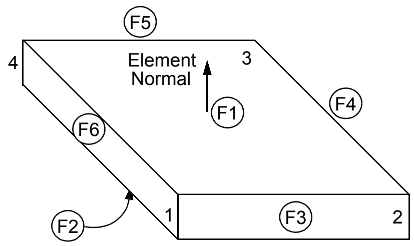
- Added *Add All Connected Elements* option to the *Pick^* menu of the standard entity selection dialog box when selecting elements.
- Added *on Element* to the *Method^* menu in the standard entity selection dialog box when selecting Coordinate Systems.
- Added *on Solid* to the *Method^* menu in the standard entity selection dialog box when selecting Regions.
- Added ability to use *Pick Query* or *Pick Front* when selecting Regions.
- Added ability double-click in the Graphics window, when not in a command, to Autoscale. See Views for details.
- Updated fast picking to improve performance when picking all entity types. It is now about 10X faster.
- Updated font selection in Graphics window to support new fonts loaded with Windows 8 and 10
- Updated icons for Properties to reflect the Property Type. If Bar/Beam Shape is defined, the icon will be shown as the selected shape (“G” icon is for General Section beam).
- Updated Property drop-down controls on many commands to automatically only show properties that are compatible with the current operation. In commands where a property can be created “on-the-fly”, it now automatically switches the active Property type to something that is compatible with the operation if the current active Property type is incompatible. Also, the width of the drop-down list will now adjust to show titles wider than the drop-down control.
- Updated usage of the mouse wheel to work on the window/pane the cursor is currently over. Allows multiple graphics views to be manipulated and panes to be scrolled without first clicking in the window to activate the window. In Windows 7 clicking in the window to activate is still required when a dialog box in not open. Only graphics windows from the current model can be manipulated.
- Updated Face Selection in all command which involve selecting Element Faces or Element Edges.

Specifying Face IDs

When selecting element faces, you will be presented with the following *Face Selection* dialog box to choose the face or faces where the pressure will be applied:



This provides six ways to select the faces. For details on how face numbers for plane and solid elements are defined, see Section 6, “Element Reference” in the *FEMAP User Guide*. Alternatively, you can simply choose the face graphically by moving the cursor near the center of the face and clicking the left mouse button. The selected face will be highlighted. When using *Adjacent Faces* or *Face ID*, if you chose an unexpected face, simply move the mouse and click again until you get the face you want. When using *Individual Faces*, any unexpectedly chosen face will need to be removed from the list of faces, otherwise it will remain selected. Also, several methods offer the option to select the *Front Face* or the *Back Face* when choosing the face of a plate element. This is strictly a way to choose a particular face without having to rotate the model.



Triangular elements do not have a face 6.

In most cases, loads on plane elements will be applied to face 1. In this case positive pressure acts in the same direction as the face normal (as determined by the right-hand rule). Conversely, if loads are applied to face 2, their positive direction will be opposite to the face normal. Therefore a positive pressure on face 2 is equivalent to a negative pressure on face 1. If you need to apply edge loads, they can be applied to faces 3 through 6 as shown. Their positive direction is inward, toward the element center.

It is possible to use a combination of different methods or the same method multiple times to select element faces. At any time, you can click the *More...* button and the faces selected by the current *Method* will be added to the “overall face selection list”. A running count of faces currently in the “overall face selection list” will appear in the upper right of the dialog box. If you have any faces selected by a *Method* and have not yet clicked the *More...* button, then switch to a different *Method*, you will be asked “Ok to Select (#) Faces from Previous Method?”. If you answer *Yes*, the faces will added to the “overall face 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*.

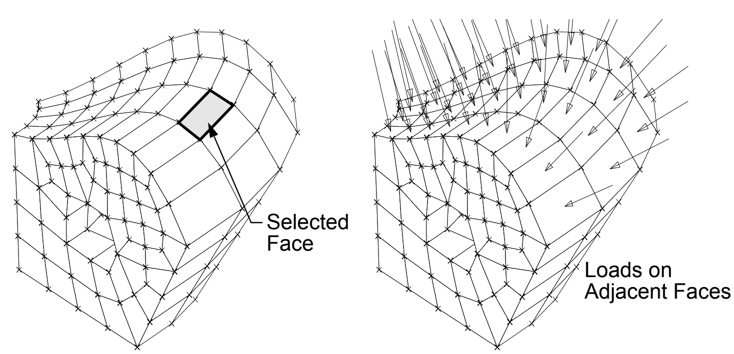
If you would like to limit which elements which will be considered when using a method which automatically determines faces (for instance, *Adjacent Faces*), click the *From (#) Elements...* button to use the standard entity selection dialog box to select elements. You may need to use the *Reset* button in the *Entity Selection* dialog box to clear the list of selected elements before choosing elements to limit face selection. You can also 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 faces will be selected by certain methods. To see a “preview” of selected element faces in the graphics window, click the *Show* icon button above the *From(#) Elements...* button and the element faces will be highlighted. When done with the “preview”, simply click the *Show* icon button again.

Finally, you can remove all faces from the “overall face selection list” by clicking the *Clear List* icon button next to the *Show* icon button.

Choosing Adjacent Faces

The most powerful method for choosing faces, especially for complex solid and planar element models, is *Adjacent Faces*, which is the default. You choose just one initial face (and the associated element ID). This can be done very easily by graphically selecting the face. You then specify a tolerance angle. FEMAP will search all selected elements for faces that are connected to the face that you chose and that are within the specified tolerance from being coplanar (colinear for planar elements) with an already selected face. This can be used to find all faces on an outer surface (or edge) of a solid (or planar) - regardless of the shape. By selecting the option *Matching Normals Only* you can further limit the faces selected by allowing only elements with matching normals to be selected.



In the picture above, loads could have been applied to all exterior faces, including those inside the hole, by choosing a tolerance greater than 90 degrees. Loads could have been applied just in the hole by selecting a face inside the hole and specifying a fairly low tolerance. You have the option to select the *Front Face* or the *Back Face* when choosing the face of a plate element. Again, this is strictly a way to choose a particular face without having to rotate the model.

Choosing Individual Faces

This method simply allows you to choose individual element faces one at a time, by graphically selecting faces or by entering an *Element* ID and *Face* ID, then clicking *Add to List* icon button. Faces can be removed from the list by highlighting any number of faces in the list, then clicking the *Remove from List* icon button. When a face is highlighted in the list, the face will also be highlighted in the graphics window. Similar to *Adjacent Faces*, the option to select *Front Face* or *Back Face* is also available.

Choosing Faces Near a Surface

If you have used geometry to define your elements, or if you just have surfaces in your model, you can apply loads to element faces which are close to a selected surface. When you choose *Near Surface*, you must also choose a surface and specify a tolerance. Loads will be applied to the faces of the selected elements that are closer than your specified tolerance from the surface. When *Element Faces* is selected, this method can only be used to apply pressure to Face 1 of planar elements. When *Element Edges* is selected, this method will only locate “edge faces”, which may not be valid when applying pressure loads for use in some analysis solvers.

Choosing Faces Near a Plane

The *Near Coordinates* method is very similar to *Near Surface*. Instead of specifying a surface, however, you choose a coordinate system, direction and position. This defines a planar surface, which is used along with the tolerance to find the closest faces. Again, there is an option to have it locate *Element Faces* or *Element Edges*.

Choosing Faces using a single Face ID

The *Face ID* method simply allows you to choose the ID of a single face. While this method is easy to understand, it has the disadvantage of applying the loads to the same face number on all selected elements. If the elements where you need to apply loads are oriented randomly, this method is not very effective. You will either need to use one of the other methods, or in some cases you can reorient the elements (see Section 4.8.3.12, “Modify, Update Elements, Reverse/Orient First Edge...”). Similar to *Adjacent Faces* and *Individual Faces,* the option to select *Front Face* or *Back Face* is also available.

Choosing Faces Model Free Faces

The *Model Free Faces* method simply applies the load to every “free element face” in your model, when set to *Element Faces*, while it will attempt to apply loads to all “free element edges” when set to *Element Edges*.

For more information about determining “Free Faces” see Section 7.6.3, “Free Face”.

Menu

- Added *Tools, Connection Editor* and *Tools, Toolbars, Draw/Erase* commands to the *Tools* menu. See Tools section for more information.
- Added *Mesh, Editing, Element Refine* command to the *Mesh* menu. See Meshing section for more information.
- Added *Modify, Renumber, All* command to the *Modify* menu. See Renumbering section for more information.
- Removed five commands from the *List, Output...* menu (*Force Balance to Data Table*, *Force Balance Interface Load*, *Force Balance Interface Load to Data Table*, *Freebody Nodal Summations*, and *Freebody Nodal Summations to Data Table*). The functionality from these commands has been consolidated into the *List, Output, Force Balance* command, which has been overhauled. See Listing section for more information.
- Added *Group, Operations, Generate Visible*; *Group, Operations, Generate Elem Shape*; *Group, Region, on Solid*; and *Group, Coord Sys, on Element* commands to the *Group* menu. See Groups and Layers section for more information.
- Added modified, and removed several commands on the *View, Rotate...* menu. Modified *Rotate Around View Center* to be *Rotate About View Center*. Consolidated *Rotate Around Rotation Center* and *Rotation Center* into *Rotate About Rotation Center*. The *Model Axes* toggle is now 2 commands, *Rotate Around View Axes* and *Rotate Around Model Axes*. Consolidated *Rotate Around Rotation Axis* and *Rotation Axis* into *Rotate Around Vector*. The *Single Axis* toggle is now the *Single Axis Rotation* toggle. Added *Rotate Around Coordinate System*;

Roll-Thru; and *Advanced Rotate* commands. Finally, *Spin* has been changed to *Spin Around Vector*. See Views section for more information.

- Added *View, Align By, Eye and Directions* to the View menu. See Views section for more information.
- Changed *Rotate View About...* sub-menu to *Rotate View...* on the Quick Access menu (right-mouse click in Graphics window). The commands on the *View Rotate...* menu are now *Rotate About View Center*, *Rotate About Rotation Center...*, *Rotate Around View Axes*, *Rotate Around Model Axes*, *Rotate Around Coordinate System...*, *Rotate Around Vector...*, *Roll-Thru*, *Advanced Rotate...*, *Single Axis Rotation*, and *Set View Center*. See Views section for more information.

Toolbars

- The *View-Simple* Toolbar is now the default Toolbar for manipulating views. The *View Toolbar* still exists and can be turned on using the various methods available for displaying toolbars.
- Added or modified the commands on the *View and Rotation Center* icon menu on the *View* and *View-Simple* Toolbars. The available commands are *Rotate About View Center*, *Rotate About Rotation Center...*, *Rotate Around View Axes*, *Rotate Around Model Axes*, *Rotate Around Coordinate System...*, *Rotate Around Vector...*, *Roll-Thru*, *Advanced Rotate...*, *Single Axis Rotation*, and *Set View Center*. See Views section for more information.
- Added *Visibility* commands to the context-sensitive menu for elements when element is the active entity in the *Select* Toolbar. This includes the *Hide Individual Elements...* command which controls visibility of individual elements, which was not available in previous versions.
- Added *Draw/Erase* Toolbar. See Tools section for more information.
- Added *Select a Contour Arrow View* icon to the *Post* Toolbar, which will set *Contour Style* to *Arrow.* Also, the icons for *Deformed Style* (None, Deform, Animate) and *Contour Style* (None, Contour, Criteria, Arrow) will now highlight to indicate the current mode.
- Added the *User Tools* menu to *Custom and User Tools* Toolbar (formally the *Custom Tools* Toolbar). The *User Tools* menu works exactly like the *Custom Tools* menu, but uses a different directory.

Model Info tree

- Updated functionality throughout the *Model Info* tree to automatically select all items in a particular branch, under certain conditions, even if some entities are not in the tree because the number of entities exceeds the value set for *Max Entities* on the *User Interface* tab of *File, Preferences*. Specifically, if you choose a command from a context-sensitive menu while selecting an entity header (for instance, *Properties*), all entities will be chosen. Also, if using the “Visibility check box” context-sensitive menu, the *Show Selected Only*, *Show All*, *Hide All* and *Show/Hide Reverse* commands will change visibility for all entities, not just the ones currently shown in the *Model Info* tree.
- Added *Select Show Only...* and *Select to Hide...* commands to the “Visibility check box” context-sensitive menu for *Coordinate Systems*, *Geometry*, *Regions*, *Connectors*, *Aero Panels/Bodies*, *Aero Splines*, *Aero Control Surfaces*, *Materials*, and *Properties*. In all cases, the selected command displays the standard entity selection dialog box which is used to select entities of the appropriate entity type to “Show Only” or “Hide”, respectively.
- Added *Show All Individual Elements* command to the “Visibility check box” context-sensitive menu for *Elements, By Type* and *Elements, By Shape*, which simply sets the visibility of all individual elements to “visible”.
- Added *Hide Individual Elements* command to the “Visibility check box” context-sensitive menu for *Elements, By Type* and *Elements, By Shape*, which displays the standard entity selection dialog box to select individual elements to “hide”.
- Added *Plot Study* sub-menu to the context-sensitive menu for *Analysis Study.*

The commands on this sub-menu allow you to create a *Chart* called “Quick Plot” of “nodal results vs Output Set” in the *Charting* dockable pane. These commands are only available when multiple output sets reside in a *Study* and the *Charting* dockable pane is open. Only *Translations* (*Total*, *X*, *Y*, or *Z*), *Accelerations* (*Total*, *X*, *Y*, or *Z*), and *Nodal Temperatures* are currently available.

To create a "Quick Plot", simply choose a type of nodal result from the sub-menu then select any number of nodes. A new *Data Series* will be created for each selected node.

If there is a "check mark" next to *Plot vs Set Value* near the bottom of the sub-menu, then the X-Axis will use the *Set Value* of each Output Set in the Study, otherwise the X-Axis will use the *Set ID*.

If there is a "check mark" next to *Clear Quick Plot* at the bottom of the sub-menu, then any *Data Series* currently plotted in the "Quick Plot" *Chart* will be removed, and only the *Data Series* most recently created by a *Plot Study* command will be visible. Otherwise, the newly created *Data Series* will simply be added to the "Quick Plot" *Chart*.

Meshing Toolbox

- Added functionality which tracks "Mesh Points" currently defined on surfaces, then automatically reassigns the mesh points to the appropriate surface when geometry is modified. Reassignment of mesh points only occurs when geometry is modified using commands in the various tools of the *Meshing Toolbox*, not when using commands on the *Geometry* menu.
- Added *Max Quads* option to *Mesh Surface* tool. See Meshing for details.
- Added *Combine Surfaces* to *Curve Options* when *Feature Type* is set to *Curve* in the *Feature Removal* tool.

Curve Options - When attempting to remove curves, choose from *Basic*, *Aggressive*, or *Combine Surfaces*.

Basic (default) - uses features of the Geometry, Solid, Cleanup command to try to remove redundant curves.

Aggressive - uses functionality from the "Parasolid Bodyshop" to try and "compress" very small curves out of geometry by bringing the curve's endpoints together, then "healing" the geometry to maintain valid geometry. It is quite possible to change the topology of the geometry when using this option.

Combine Surfaces - uses the Parasolid kernel to calculate a new surface through the original two surfaces, which share the selected curve. If successful, all the curves sharing the original two surfaces are removed, as they become redundant. This option will also remove any redundant points on the ends of the removed curves

- Added ability to select *Point* as a *Feature Type* to remove in the *Feature Removal* tool. When used, will attempt to remove any selected points which are redundant.

Note: When attempting to remove multiple points, all of which were selected at the same time, the tool is designed to not remove certain points which are shared by curves with a large difference in curvature. There may be instances where the curvature is not really that different, for instance when a point is shared by two splines, but the tool will not remove that point due to this designed limitation. When only a single point is selected, the tool uses a more aggressive approach, which may allow the tool to remove these points, but only when selected one-by-one.

- Added option to choose between *Femap* and *Nastran* element quality checks to the *Surface Mesh Quality* tool. When *Quality Source* is set to *Nastran*, *Quality Type* can be set to *Quad Skew*, *Quad Taper*, *Quad Warp*, *Quad IAMin*, *Quad IAMax*, *Quad AR*, *Tria Skew*, *Tria IAMax*, *Tria AR*. Also, element quality is now only shown on elements which are currently visible.
- Added option to choose between *Femap* and *Nastran* element quality checks when *Search For* is set to *Elements* and *Search Method* is set to *Quality* in the *Locator*. When *Quality Source* is set to *Nastran*, *Quality Type* can be set to *Quad Skew*, *Quad Taper*, *Quad Warp*, *Quad IAMin*, *Quad IAMax*, *Quad AR*, *Tria Skew*, *Tria IAMax*, *Tria AR*.

PostProcessing Toolbox

- Updated the default behavior of the *PostProcessing* toolbox to use the *Expand Active Tool Only* option.
- Updated the behavior of the icons used for listing in the *Freebody* tool to use the functionality of the updated *List, Output, Force Balance* command. Certain options in the *List Force Balance* dialog box will be preselected based on the *Display Mode* of the freebody entity currently active in the *Freebody* tool. Also, the *List Current Freebody to Data Table* and *List Current Freebody Summation to Data Table* icons have been removed. See Listing for more information.
- Updated the *Contour* tool when *Style* is set to *Contour Arrow* to provide access to the new capabilities when displaying an "arrow plot". Previously, all selection of output data was done by clicking the ">>>" button next to *Contour Vectors*. Here is a summary of the options which have been added or modified:

Select Arrows from Contour Vector - When this option is enabled, the *Arrow Type* is automatically selected and the *X/Y/Z Arrow Display/Vector Select* fields will also be automatically populated with Output Vectors, wen appropriate. For example, if *Output Vector* in the *Results* section is set to "7020..Plate Top X Normal Stress", "7021..Plate Top Y Normal Stress", or "7023..Plate Top XY Shear Stress", then the *Arrow Type* will be automatically set to *2D Tensor Plot*, with *X Arrow Display/Vector Select* set to "7020..Plate Top X Normal Stress", *Y Arrow Display/Vector Select* set to "7021..Plate Top Y Normal Stress", and *Z Arrow Display/Vector Select* set to "7023..Plate Top XY Shear Stress". When disabled, *Arrow Type* and *X/Y/Z Arrow Display/Vector Select* become available for manual specification.

Arrows - The options which are visible and/or available for editing in this section depend on a number of other settings for *Contour Arrows*.

Arrow Type - There are 4 options for *Arrow Type: Single Value, 2D Components*, *3D Components*, and *2D Tensor Plot*. Depending on the *Arrow Type*, the number of *X/Y/Z Arrow Display/Vector Select* fields changes. For Example, when set to *Single Value*, only the *X Arrow Display/Vector Select* field is visible. If set to *3 Components*, all three X/Y/Z Arrow Display/Vector Select fields will be visible. This option can only be modified by the user when the *Select Arrows from Contour Vector* option in disabled.

X/Y/Z Arrow Display/Vector Select - At all times, the check boxes in the fields which are currently visible are available to control the display of each individual arrow. The *Arrow Type* option controls which of these fields are visible. These fields can only be modified when the *Select Arrows from Contour Vector* option in disabled.

Arrow Color - A color can be specified for each individual arrow, unless *Use Contour Colors* option is enabled.

Note: The *Use Contour Colors* option is automatically enabled for all *Arrow Type* options except *2D Tensor Plot* when the *Auto* option is enabled for *Arrow Head and Color.*

***Advanced Options* (>>> Button)** - Button displays the *Advanced Contour Arrow Options* dialog box.

Arrow Display Mode - Allows you to select how the Arrows will be displayed. The default setting for this option is *Automatic*, which automatically selects an *Arrow Display Mode* based on the *Arrow Type* option and the selected output vector(s). *Automatic* will display all "Total Vectors" as resultants, while all others are displayed as components. Additional options include:

Components - displays individual arrows for *2D Components* (*X* and *Y*) and *3D Components* (*X*, *Y*, and *Z*).

Resultant - displays a single resultant vector for *2D Components* or *3D Components*.

Shear (XY)/Axial (Z) - displays *X* and *Y* arrows as resultant (*"Shear"*) and the Z arrow as a component (*"Axial"*).

Shear (XZ)/Axial (Y) - displays *X* and *Z* arrows as resultant (*"Shear"*) and the Y arrow as a component (*"Axial"*).

Shear (YZ)/Axial (X) - displays *Y* and *Z* arrows as resultant (*"Shear"*) and the X arrow as a component (*"Axial"*).

Arrow Direction - Allows you to specify a vector as a display direction, and only for *Arrow Type* is set to *Single Value*.

Transform - Allows Transformation of Contour Output Vector(s). Available options depend on type of output data being transformed. Click icon button to access *Current Output Orientation* dialog box. See the Contour Output Vector Transformation portion of Section 8.2.2.2, "Selecting Data for a Deformed or Contour Style" for more info.

Nodal Vector Output - Nodal Vectors may be transformed using the *Nodal Output Coordinate System* of each node or into any *Coordinate System* in the model.

Plate Force/Stress/Strain - Plate Force/Stress/Strain and Laminate Stress/Strain may be transformed using the *Material Angle* of the Elements (material angle set using the *Modify, Update Elements, Material Angle* command), *Along a Specified Vector* (*Along Vector* field and icon button will appear for specification of vector), or into the X, Y, or Z axis of any *Coordinate System* in the model (*Axis* field will appear for to selection of *X*, *Y*, or *Z* axis).

Solid Stress/Strain - Solid Stress/Strain may be transformed using the Material Direction of the Elements or into any *Coordinate System* in the model.

Complex Results Mode - Only available when the selected Output Set contains complex output. Allows you to set overall "Model Option(s)" or view-specific "override" values used to convert complex data to real data, on-the-fly, when post-processing complex results. See Complex Results... in Section 8.2.2.2, "Selecting Data for a Deformed or Contour Style". The *Complex Model Options* icon button can be used to set the "Model Option(s)".

Phase Angle - A value for phase angle can be entered directly in whole degrees or a slider control can be used to dynamically set the value from 0 to 360.

Arrow Head and Color - When set to *Auto*, automatically sets values for *Arrow Head*, *Location*, and *Use Contour Colors*, depending on other options specified for *Contour Arrows*. When not set to Auto, these options can be set individually,

Arrow Head - Choose from *Double-Ended* (arrowheads on both ends), *Single* (arrowhead on one end), *None*, or *Moment* (two arrowheads on one end).

Location - Choose between *At Tail* (depending on type of output, Arrow extends out from node or element centroid) or *Center* (Arrow centered at node or element centroid).

Use Contour Colors - When on, Arrows are displayed using "Contour Colors" and the *Contour Legend* is also visible. When off, Arrows are displayed using individual *Arrow Color* and *Contour Legend* is not visible.

Solid Arrows - When on, Arrows are displayed using solid representations. When off, Arrows are wireframe.

Arrow Length - Allows you to specify a length for all Contour Arrows (default is 100.0).

Scale by Magnitude - When on, Contour Arrows are displayed based on magnitude, with the Maximum value displayed using 1.0**Arrow Length* and Minimum value displayed as *Min Length* (0.25 default)**Arrow Length*.

Min Vector Magnitude - When on, Arrows with a magnitude below this value will not be displayed.

Arrow Labels - Choose *On*, *Off*, or *Exponent*. *Label Digits* value sets number of displayed digits, 0 is "Auto".

Charting pane

- Added *Scale Override* option to *Chart Settings* tab of *Charting* dialog box. When enabled, the Y values of all *Data Series* currently displayed in a *Chart* will be scaled by the *Scale Override* value, not the value specified for *Scale* on the individual *Data Series*. If a *Chart* does not have this option enabled, then the value specified for *Scale* on the individual *Data Series* is used.
- Added *Study* drop-down on the *Vector vs. Output Set* and *Vector vs. Vector* tabs in the *Chart Data Series* dialog box, which allows you to specify a range of output sets by selecting an Analysis Study.

Entity Editor

- Added appropriate NX Nastran element quality values when an element is in the *Entity Editor*. For instance, if a quadrilateral element is in the *Entity Editor*, you will see *Nastran SKEW (Quad Skew)*, *Nastran IAMIN (Quad IAMIN)*, *Nastran IAMAX (Quad IAMAX)*, and *Nastran AR (Quad AR)*, while the value for *Quad Warp* is the same as the *Nastran Warping* FEMAP element quality check and the value for *Quad Taper* is the same as the *Alt Taper* FEMAP element quality check, thus they are not shown individually.

Connection Editor

- The *Connection Editor* dockable pane was added for 11.3. See Tools section for full description.

Data Table

- Added additional options to the *Data Table* filter, specifically to filter entities by *Text*. Previously you could only filter Text as *Contains*. Now, you can choose *Contains*, *Not Contains*, *Equals,* or *Not Equals*, with *Contains* and *Not Contains* only needing to match a portion, while *Equals* and *Not Equals* need an exact match. All options are not case-sensitive.
- Updated the *Show When Selected* capability for Connectors to highlight both the Connector and associated Connection Regions, which now matches the behavior of the *Model Info* tree.

Geometry

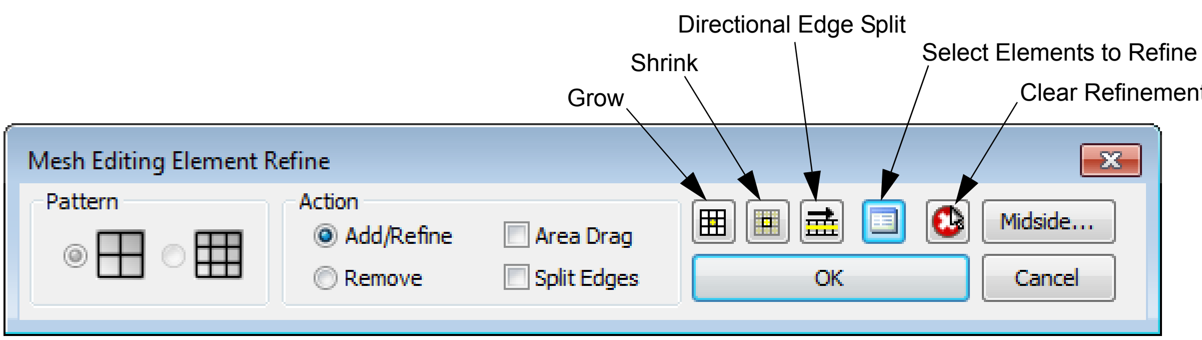
- Added *Plane to Plane* method for *Modify, Align...* commands to align geometry (*Point, Curve*, *Surface*, *Solid*, and *Volume*). When this method is used, the behavior is similar to *Between Coordinate Systems*, only each specified plane (X and Y axes) and each plane's normal direction (Z axis) are used to determine XYZ axes.

- Updated *Geometry, Solid Cleanup* command, which allows the user to *Enable Advanced Cleanup*, choose categories of “Advanced Cleanup” to attempt, and/or click the *Advanced Cleanup Options...* button to display a dialog box to select individual options in four different categories. Also, added option to *Remove Gashes*.
- Added *Angle Tolerance* to *Geometry, Surface, From Mesh* command. Controls the allowable difference, in degrees, between the “average mesh normal” at each node of the original mesh and the normal of the newly created surface, at each nodal location. Making this value larger may create geometry which is smoother, but may also cause the new surface to be further away from the original nodal locations.
- Updated commands on the *Geometry, Copy...*; *Geometry, Scale...*; *Geometry, Rotate...*; and *Geometry, Reflect...* menus to not copy any attachment or reference to boundary surfaces and/or combined curves if only the underlying geometric entities are copied, scaled, rotated, or reflected.

Meshing

- Added *Max Quads* option to *Mesh, Geometry, Surface* command. When used, the surface mesher will attempt to create as few triangular elements as possible and in certain cases, will produce a “quad only” mesh.
- Added capability to the *Mesh, Editing, Edge Split* command to select multiple nodes and automatically split between all of the element edges that they define.
- Added capability to the *Mesh, Extrude, Curve*; *Mesh, Extrude, Element*; and *Mesh, Extrude, Element Face* commands to specify the extrusion direction and number by selecting either element edges or a path of nodes. This allows you to easily extrude parts of an existing mesh to locations that match another portion of a mesh, including irregularly spaced locations.
- Added Mesh, Editing, Element Refine command.

This command allows you to dynamically “highlight” shell elements to refine using one of two patterns, “original quad element split into four quad elements” (i.e., the 1 to 4 Pattern) or “original quad element split into nine quad elements” (i.e., the 1 to 9 Pattern). Any elements surrounding the “refined elements” will be automatically split using appropriate “transition patterns” to maintain connectivity between the “newly refined mesh” and the “original mesh”. As the elements are “highlighted”, a preview of the “refined mesh” and “transition mesh” will be “painted” on the original elements. Once you are satisfied with the mesh refinement, click OK to update the original mesh. In addition, there is an option to split shell elements in two halves or into thirds by using specified vector to “split along”. Finally, any line elements sharing nodes with a refined or modified shell element will also be split appropriately.



To use the command:

1. In the *Mesh Editing Element Refine* dialog box, pick a *Pattern* for the element refinement.

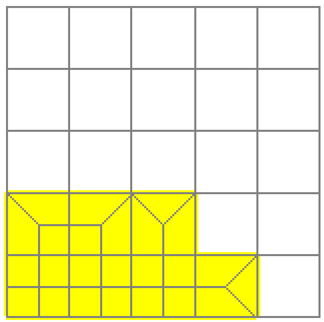
2. By default, *Action* will be set to *Add/Refine*. Move your cursor over the model, then “click and drag”. As the cursor passes over an element, the command will preview the “split” on that element, as well as “transition splits” on the surrounding elements.

3. If satisfied with the mesh refinement and transitions, simply click *OK* and the mesh will be updated. If not, change *Action* to *Remove*, then “click and drag” to begin removing elements from the refinement. If you would like to start the element selection process over from the beginning, click the *Clear Refinement* icon button, then “click and drag” to begin selecting elements to refine until satisfied, then click *OK*.

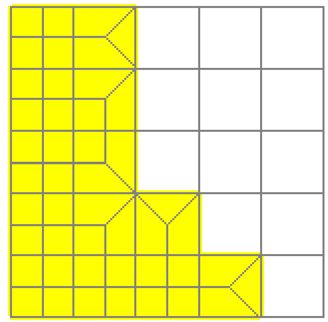
4. This command autorepeats, so you must click *Cancel* or press *Esc* key to completely exit the command.

Options on the *Mesh Editing Element Refine* dialog box include:

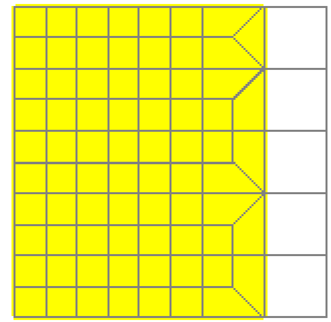
Area Drag - When this option is enabled, an attempt will be made to "expand" selection to all of the elements currently highlighted in a particular "area" of the model, when using "click and drag"



Original Refinement Area



Area Drag option disabled



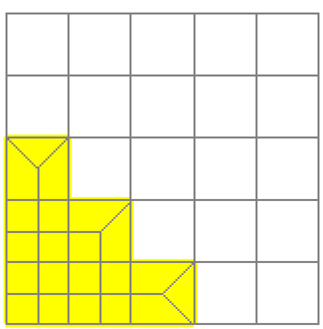
Area Drag option enabled

Edge Split - When this option is enabled, you are selecting element edges instead of element faces. This can be useful when trying to refine as few elements as possible. Also, if you click the *Select Elements to Refine* icon button when this option is enabled, the *Element Face* selection dialog box will appear instead of the standard entity selection dialog box. For more information, see Section 4.3.3.4, "Model, Load, Elemental..."

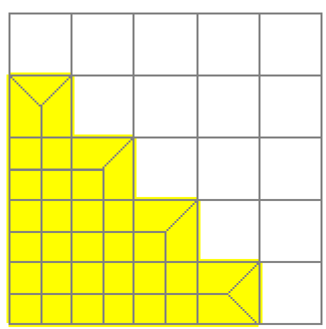
Icon buttons in the *Mesh Editing Element Refine* dialog box include:

Grow - Expands current selection area(s) by adding one layer of elements to the outside of the current selection area(s). The *Area Drag* option is not considered when using the Grow functionality.

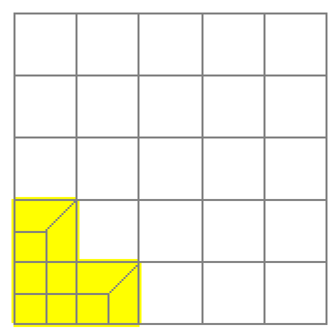
Shrink - Reduces current selection area(s) by removing one layer of elements from the outside of the current selection area(s). The *Area Drag* option is not considered when using the Grow functionality.



Original Refinement Area

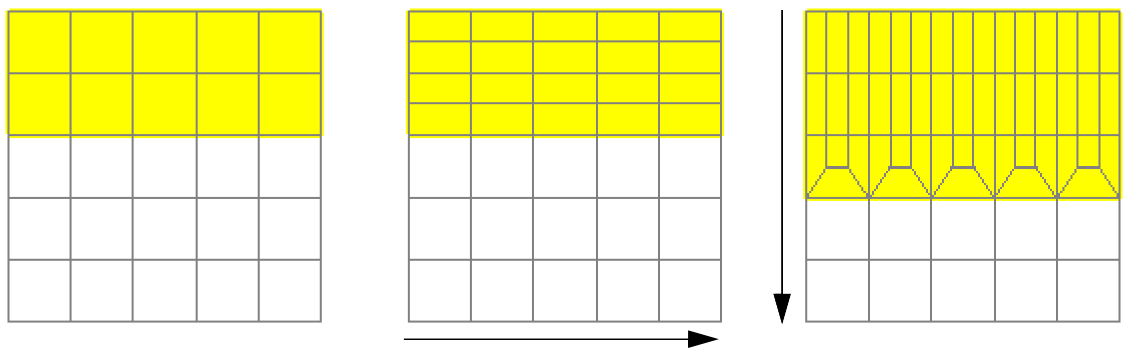


Area after single Grow



Area after single Shrink

Directional Edge Split - This icon button accesses functionality which is somewhat different than the rest of the command. When used, you will prompted to *Select Element(s) to Refine* using the standard entity selection dialog box for elements, then *Select Split Direction* by defining a vector with the standard *Vector Locate* dialog box. The number of splits for each element is determined by the *Pattern* selected in the *Mesh Editing Element Refine* dialog box, therefore each element is split into two elements when using the "1 to 4 Pattern" or three elements when using the "1 to 9 Pattern". Once the elements have been selected and the vector defined, a preview of the element splitting will be "painted" on the original elements. Similar to the rest of the command, any elements surrounding the "split elements" will be automatically split using appropriate "transition patterns" to maintain connectivity between the "newly split mesh" and the "original mesh" and will be highlighted as well. If satisfied with the mesh refinement and transitions, simply click *OK* and the mesh will be updated. If not, change *Action* to *Remove*, then "click and drag" to begin removing elements from the refinement. If you would like to start the element selection process over from the beginning, click the *Clear Refinement* icon button, then "click and drag" to begin selecting elements to refine until satisfied, then click *OK*.



Elements to be Split   Vector from Left to Right (1 to 4)   Vector from Top to Bottom (1 to 9)

Select Elements to Refine - Click this icon button to choose elements to refine using the standard entity selection dialog box. If *Split Edges* is enabled, the Face Selection dialog box will appear instead.

Clear Refinement - Clears all highlighting to allow you to start mesh refinement from the beginning, using any functionality available in the *Mesh Editing Element Refine* dialog box.

- Updated *Mesh, Editing, Edge Split* command to automatically adjust element corner thicknesses when splitting tapered planar elements.
- Updated the Quad mesher and Tri Subdivision mesher to honor the *Surface Interior Mesh Growth* factor specified in the *Automatic Mesh Sizing* dialog box.
- Updated the *Mesh, Editing, Interactive* and *Mesh, Editing, Split* commands to only merge nodes of elements that have been split or those adjacent to a split. Previously all nodes were merged.
- Updated various commands on the *Mesh, Editing...* menu to create and maintain associativity to multiple solids for nodes that lie on the boundary of adjoining solids.
- Updated the functionality of the icon buttons in dialog box of the *Mesh, Editing, Rigid Connectivity* command to simply highlight all of the nodes on the element or element instead of bringing up a single entity selection dialog box. This dialog box was also updated for the *Mesh, Connect, Rigid* command and the *Model, Load, From Freebody* command, when using *Multi-Model*.

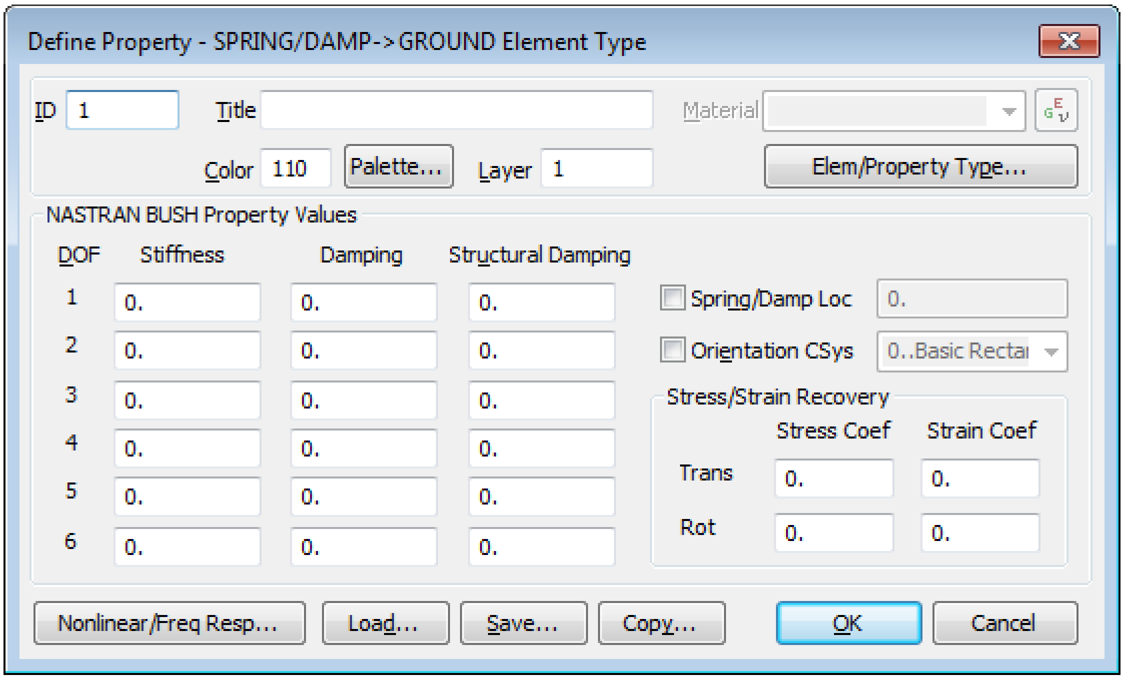
Elements

- Added *Plane to Plane* method for *Modify, Align...* commands to align nodes and elements When this method is used, 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.
- Added *Increment* and Percent methods to the *Modify, Update Elements, Adjust Plate Thickness/Offset* command. If using *Increment*, the specified value can be positive or negative and will simply be added to or subtracted from the existing thickness/offsets. If using *Percent*, the specified value can be positive or negative, must be entered as a percentage, and is used to modify the thickness/offsets by multiplying the original thickness/offsets by 1/100 of entered value, then adding the value to or subtracting the value from the original thickness/offsets.
- Added *Spring/Damper to Ground* element type. This is a single node CBUSH-Style element which connects the selected node to "Ground" (essentially, connects the selected node to a "phantom" node with the same coordinates, constrained in all 6 DOF). Stiffness, damping, structural damping, and other options are defined on the *Spring /Damper to Ground* property.
- Added *DOF Spring to Ground* element type. This is a single node spring which connects a single DOF of the selected node to "Ground" (essentially, connects the selected node to a "phantom" node with the same coordinates, constrained in all 6 DOF). Stiffness, damping, and other options are defined on the *DOF Spring to Ground* property.

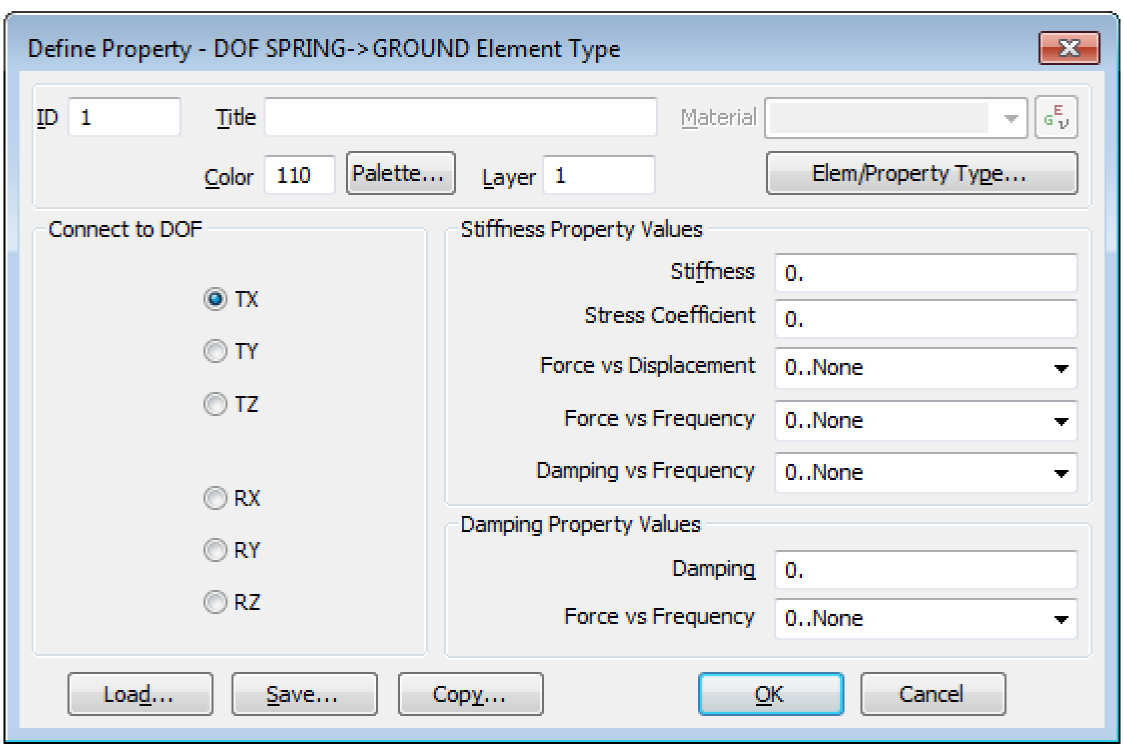
- Updated the name of the *Stiffness Matrix* element type to *General Matrix*. This type of element can now be used to include stiffness, damping, or mass between two nodes, using values in either a 6x6 or 12x12 matrix specified in the *General Matrix* property. Also, *General Matrix* elements are now shown using a “[X]” symbol.

Properties

- Added *Spring/Damper to Ground* property type which specifies *Stiffness*, *Damping*, *Structural Damping*, and other options for the *Spring/Damper to Ground* element.

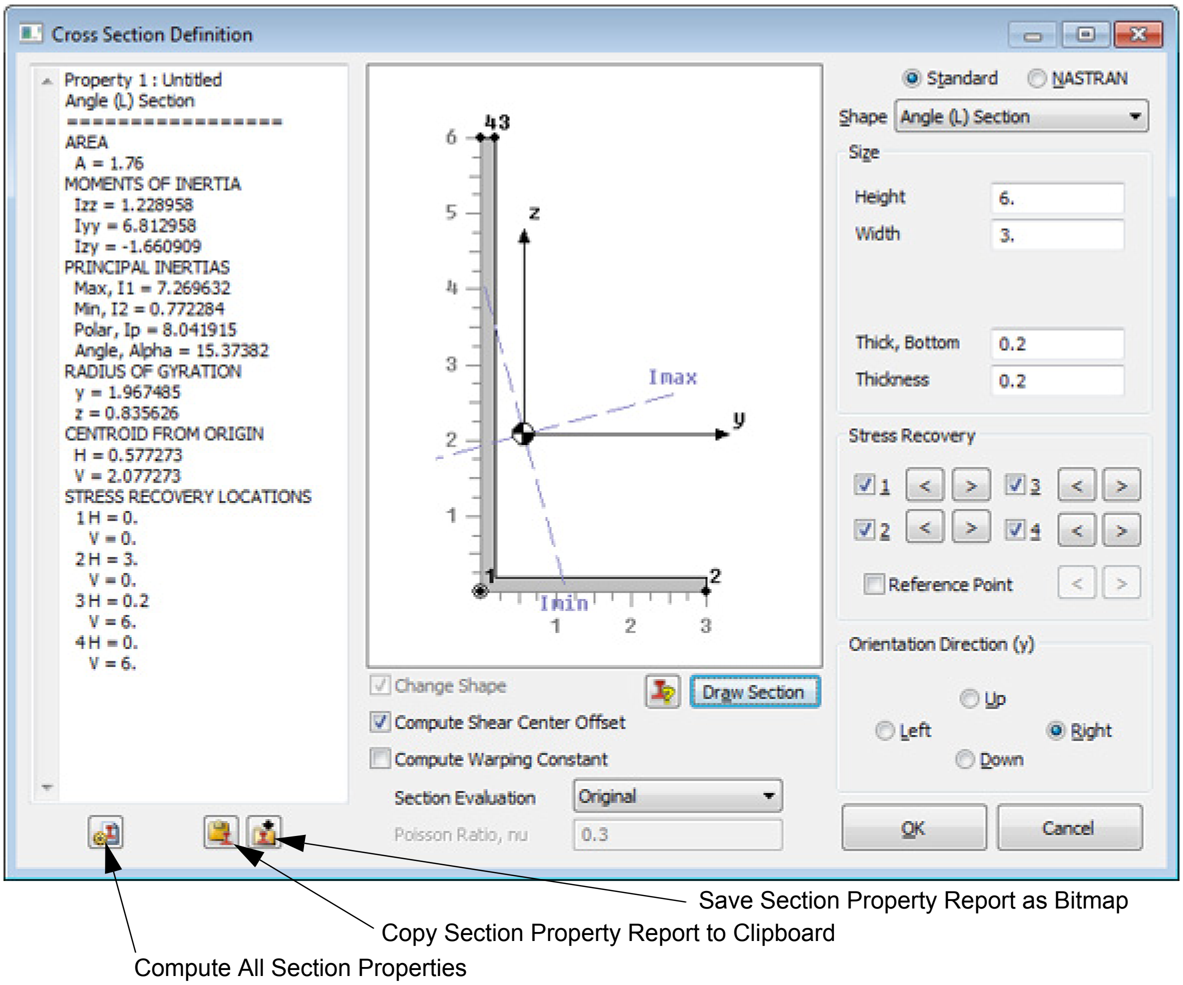


- Added *DOF Spring to Ground* property type which specifies *Stiffness*, *Damping*, *Connect to DOF*, and other options for the *Spring/Damper to Ground* element.



- Updated the name of the *Stiffness Matrix* property type to *General Matrix*. There is now a *Matrix Type* which allows the specified values to be used as a Stiffness Matrix, Damping Matrix, or Mass Matrix. Also, the values can now be entered using a 6x6 Matrix or 12x12 Matrix.
- Updated the *Cross Section Definition* dialog box, which is accessed by clicking the *Shape* button in the *Define Property...* dialog box when creating a *Bar*, *Beam*, or *Curved Beam* property. This now provides useful information to the user and this information can be copied to the Clipboard or saved to a file for use in reports.

Clicking the *Shape* button will display the *Cross Section Definition* dialog box:



This dialog box allows you to select a *Standard* or *NASTRAN* "shape" using the *Shape* drop-down, then enter the required dimensions for the selected "shape" in the *Size* section. The middle portion of the dialog box will display a "Help Diagram" showing dimensions for the selected shape until all of the required dimensions have been entered, then it will display a picture of the defined cross section. To display the "Help Diagram" again after all dimensions have been entered, click the icon button next to the *Draw Section* button, then click *Draw Section* to see the defined cross section again. The various section on the right are explained in greater detail below.

Included in the picture of the defined cross section are the *y* and *z* axes with a black and white circle, which represent the location of the neutral axis and show the current *Orientation Direction (y)*; an unfilled circle representing the Origin; points labeled 1-4, which correspond to 1-4 in the *Stress Recovery* section; dotted lines shown at a specific angle, which represent the Principal Inertias; and "rulers" for the overall height and width. If *Reference Point* is enabled in the *Stress Recovery* section, then a point labeled "R" will also appear.

The left portion of the dialog box displays a subset of "section properties" which can be simply calculated from the selected shape and entered dimensions, such as *AREA*, *MOMENTS OF INERTIA*, *PRINCIPAL INERTIAS*, and *RADIUS OF GYRATION*, along with the distance of the *CENTROID FROM ORIGIN* and the *STRESS RECOVERY LOCATIONS* expressed in terms of *H (Horizontal)* and *V (Vertical)* with regard to the Origin. To calculate additional "section properties", such as *TORSIONAL CONSTANT*, *SHEAR AREA*, *OUTER PERIMETER*, *SHEAR CEN-*

TER FROM ORIGIN, and *SHEAR CENTER FROM CENTROID*, click the *Compute All Section Properties* icon button in the lower left of the dialog box. In addition, the cross section dimensions will also appear with the "section properties" and a "pink diamond" representing the location of the shear center will appear in the picture.

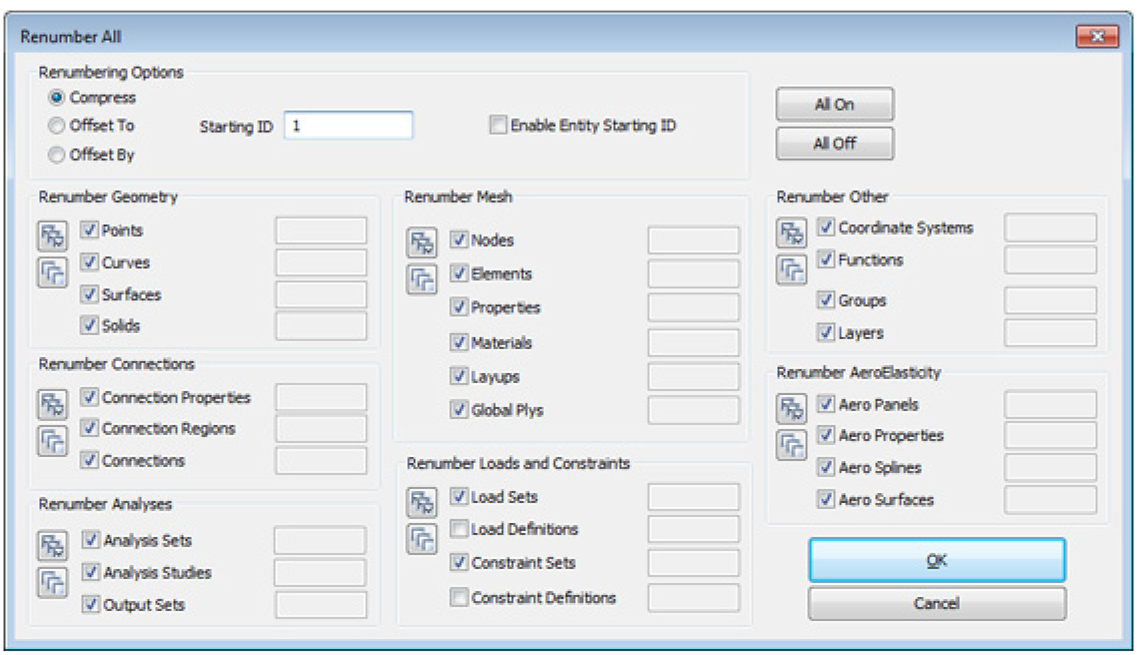
The *Copy Section Property Report to Clipboard* icon button will copy the picture of the defined cross section and the section properties currently displayed to the clipboard, while the *Save Section Property Report as Bitmap* icon button will use save the same information as a Bitmap file. It may be beneficial to use the *Compute All Section Properties* icon button before using either of these commands to maximize the data.

Loads and Constraints

- Added ability to modify the color, modify the layer, scale the values, change the function ID, or delete loads of specified type(s) in the active Load Set, all Load Sets, or any number of selected Load Sets. This functionality is found in the *Modify, Color, Load*; *Modify, Layer, Load*; *Modify, Update Other, Scale Load*; *Modify, Update Other, Load Function ID*; and *Delete, Model, Load - Individual* commands. Also, updated *Modify, Update Other, Load Phase* and *Delete, Model, Load - Body* commands to allow selection of any number of Load Sets.
- Added ability to modify the color, modify the layer, or delete constraints of specified type(s) in the active Constraint Set, all Constraint Sets, or any number of selected Constraint Sets. This functionality is found in the *Modify, Color, Constraint*; *Modify, Layer, Constraint*; and *Delete, Model, Constraint - Individual* commands.
- Added ability to specify "non-zero constraints" using the *Model, Constraint, Nodal* command.
- Added a warning message/question prompting the user to save the model if geometric constraints are translated that cause the nodal output coordinate systems to be updated. This can occur when exporting/analyzing Nastran, ANSYS, and ABAQUS models.
- Updated *Modify, Edit, Load - Definition* to issue a warning message to inform when the user is editing a Load Definition which contains non-uniform load values. Also, updated Modify, Edit, Constraint Definition to issue a warning when editing a Constraint Definition which has non-uniform degrees of freedom specified.
- Updated *Model, Load...* commands to support creating corner pressures when using *Variable* or *Data Surface.*

Renumbering

- Added *Modify, Renumber, All* command. This command is designed to renumber all individual entities of each entity type currently in the model via a single *Renumber All* dialog box:



For simplicity, the *Renumber All* dialog box offers only a subset of functionality available in the individual *Modify, Renumber...* commands for each entity type and always operates on all entities of the selected entity type(s).

By default, the command is designed to "compress" the IDs of all entity types into a continuous range for each entity type, all beginning with a single *Starting ID*, but also offers options to accomplish more complicated renumbering tasks.

Renumbering Options

This section offers a choice of 3 different renumbering "strategies" (*Compress, Offset To*, and *Offset By*), the ability to specify a *Starting ID*, and a toggle to *Enable Entity Starting ID*.

Compress - when selected, which is the default, the IDs of the selected entity types will be "compressed" by keeping the original entity order and removing any "gaps" between IDs for each selected entity type. The IDs will change, but not the relative sequence of entities within your model.

Offset To - when selected, will simply change the lowest ID value for each selected entity type to the *Starting ID*. It will then add the constant difference between the original lowest ID and the *Starting ID* 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 changing all IDs from 1-1000 to 10,001-11,000 by using a *Starting ID* of 10,001)

Offset By - when selected, *Starting ID* will change to *ID Offset* and will simply add the value of the *ID Offset* to the original ID of each entity. 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 changing all IDs from 1-1000 to 10,001-11,000 by using a *ID Offset* of 10,000).

Starting ID/ID Offset - used to enter a value to use as a Starting ID when *Renumbering Options* is set to *Compress* or *Offset To*, or an *ID Offset* when *Renumbering Options* is set to *Offset By*. If the Enable Entity Starting ID toggle is checked, this value will be used to populate the ID field next to all of the entity types currently selected in the *Renumber All* dialog box.

Enable Entity Starting ID - when this option is enabled, the individual ID fields for each entity type currently selected in the *Renumber All* dialog box will become available and be populated with the currently specified value for *Starting ID/ID Offset*.

Renumber sections and All On/All Off buttons

The bottom portion of the *Renumber All* dialog box is divided into 7 different sections, with each section representing a group of entity types which a user may want to renumber concurrently.

Each individual entity type can be toggled on/off using the check box to the left of the entity type. By default, all selected entity types will use the *Starting ID/ID Offset* in junction with the selected renumbering option. If *Enable Entity Starting ID* option is enabled, then an individual ID may be entered for each selected entity type and will be interpreted appropriately based on the selected renumbering option.

In each section, there are *Select All* and *Select None* icon buttons which can be used to toggle all entity types on/off in a that section. In addition, the *All On* and *All Off* buttons in the upper right of the *Renumber All* dialog box can be used to toggle all entity types on/off.

Listing

- Updated the *List, Model, Load - Individual* command to allow you to list loads in the Active Load Set, all Load Sets, or any number of selected load sets. The same capability was add for the *List, Model, Constraint - Individual* command, only you choose to list from the active Constraint Set, all Constraint Sets, or any number of selected Constraint Sets.
- Updated *List, Output, Contoured Results to Data Table* command when *Contour Style* is set to *Arrow*.

When the *Select Output from Contour Vector* option is enabled, lists the "Element ID" or "Node ID" in the first column, depending on the type of output being displayed. Then, depending on the option set for *Arrow Type*, up to 3 additional columns containing output values will be listed. The additional columns for each *Arrow Type*:

Single Value - One column with values from the output vector selected for *X Arrow Display/Vector Select*.

2D Components - Two columns with values from the output vectors selected for *X Arrow Display/Vector Select* and *Y Arrow Display/Vector Select*.

3D Components - Three columns with values from the output vectors selected for *X Arrow Display/Vector Select*, *Y Arrow Display/Vector Select*, and *Z Arrow Display/Vector Select*.

2D Tensor Plot - Three columns with values from the output vectors selected for *X Arrow Display/Vector Select*, *Y Arrow Display/Vector Select*, and *XY Arrow Display/Vector Select*.

In addition, any vector(s) displayed as a “Resultant” will only list a “Resultant” column, while any vector(s) displayed as “Axial/Shear” will list both an “Axial” column and a combined “Shear” column.

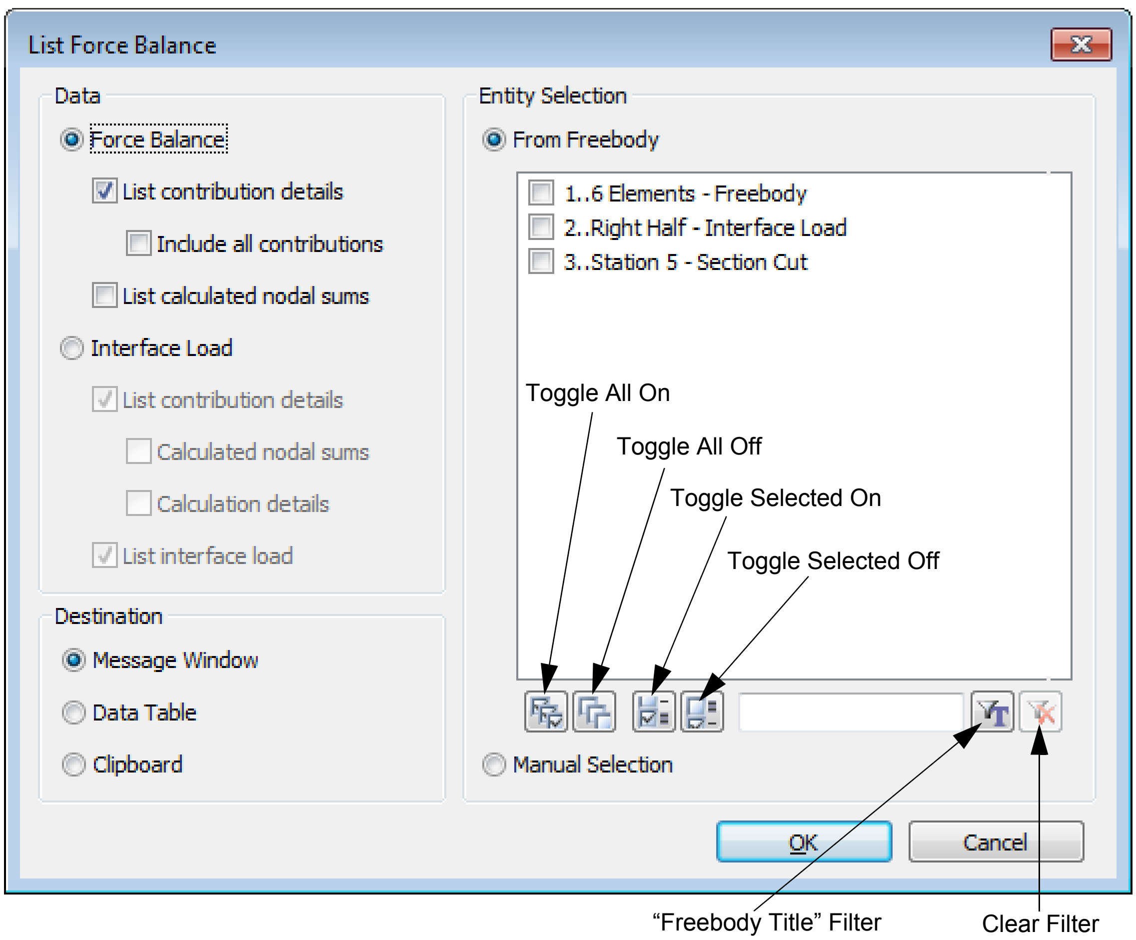
When the *Select Output from Contour Vector* option is disabled, then the output will be listed using a “Element ID” or “Node ID”, a “Vector” (1, 2, or 3), a “Value”, and “X” (always), “Y” (when output vector has 2 components), and “Z” (when output vector has 3 components).

- Updated the *List, Output, Force Balance* command to consolidate functionality from various commands previously on the *List, Output...* menu (*Force Balance to Data Table*, *Force Balance Interface Load*, *Force Balance Interface Load to Data Table*, *Freebody Nodal Summations*, and *Freebody Nodal Summations to Data Table*).

This command allows you to list a *Force Balance* or an *Interface Load* to a specified *Destination,* by allowing you to select existing Freebody entities from the *From Freebody* list, or by selecting desired entities using a *Manual Selection* mode.

Note: This command is designed to be used in models where grid point force results data has been recovered from Nastran. If the grid point force data is not available, then the listing will only consist of data which was requested and recovered, so typically this would be forces and moments from applied loads, single point constraints, multi-point constraints, contact, and/or glued contact.

All modes and options are specified using the *List Force Balance* dialog box:



The dialog box has three sections: *Data*, *Destination,* and *Entity Selection*, which are all used concurrently. This allows you the flexibility to pick and choose various modes and options to produce the desired listing.

For example, you can have an *Interface Load* listing of an existing *Freebody* entity sent to the *Data Table*. This can be done with minimal user input by simply selecting an existing Freebody in the *From Freebody* list. On the other hand, you may want to pick and choose the Contributions, Nodes, Elements, etc. for a unique *Force Balance* listing sent directly to the *Clipboard*. If this is the case, you would want to use the *Manual Selection* option in the *Entity Selection* section of the dialog box.

Listing

When *Destination* is set to *Messages Window*, a "Title Block" will appear above the listing for each selected *Freebody ID*/*Output Set* combination (*Entity Selection* set to *From Freebody*) or simply for each *Output Set* (*Entity Selection* set to *Manual Selection*).

When using *Interface Load*, additional information about the *Components included in summation* and *Contributions included in summation* will appear above the individual contributions, along with the *Summation about* location. In addition, a ***TOTAL SUMMATION*** row will appear after the individual contributions.

When *Destination* is set to *Data Table* or *Clipboard*, no "title block" or additional information is included and the ***TOTAL SUMMATION*** row appears above the individual contributions for each *Node ID*, instead of at the end.

Each listing always contains these 8 columns: *Node ID*, *Source*, *Fx*, *Fy*, *Fz*, *Mx*, *My*, and *Mz*.

Node ID - This column contains the "Node ID" of each contribution. All contributions for a given *Node ID* will be always be listed together and in chronological order. For example, all contributions for *Node ID* 1 would be listed, then all the contributions for *Node ID* 2, etc.

Source - This column contains the "source" of each row, whether it be a contribution or calculated nodal sum:

APP-LOAD - Contribution from an Applied Load

F-OF-SPC - Contribution from a Single-Point Constraint

F-OF-MPC - Contribution from a Multi-Point Constraint

ELEM # (P) - Contribution from a Peripheral Element

ELEM # (F) - Contribution from a Freebody Element

CONTACT - Contribution from Contact

GLUE - Contribution from Glued Contact

TOTALS - Contribution from Nodal Summation (*Reverse Value* simply reverses sign of values)

SUM - Calculated Nodal Sum (only appears when option is selected in *List Force Balance* dialog box)

Fx, **Fy**, **Fz** - These columns contain values for X, Y, and Z Force, respectively.

Mx, **My**, **Mz** - These columns contain values for X, Y, and Z Moment, respectively.

When using *Interface Load* with the *Calculation Details* option enabled, these 9 additional columns will appear: *d1*, *d2*, *d3*, *Fx'*, *Fy'*, *Fz'*, *Mx'*, *My'*, and *Mz'*.

d1, **d2**, **d3** - These columns contain the distances from each Node ID to the "Total Summation" location.

Fx', **Fy'**, **Fz'** - These columns contain values for X, Y, and Z Force used in the "Total Summation" calculation.

Mx, **My**, **Mz** - These columns contain values for X, Y, and Z Moment, used in the "Total Summation" calculation.

Data

This section allows you to choose if you are listing a *Force Balance* or an *Interface Load*. Each "mode" offers a few options which can be toggled on/off to provide more or less detail, as needed. Unless only one Output Set exists in the model, you will always be prompted to select Output Sets.

Force Balance - Lists force and moment contributions acting on a set of nodes for each selected Output Set. Contributions and nodes are determined automatically from the selected Freebody entity (*Entity Selection* set to *From Freebody*) or selected by the user (*Entity Selection* set to *Manual Selection*).

There are 3 options when listing the *Force Balance*:

List Contribution Details - When "on", lists each selected contribution at each Node ID. On by default.

Include All Contributions - Only available when *List Contribution Details* is turned "on". When "on", all contributions of every type are listed, otherwise, only the selected contributions will be listed. Off by default.

List Calculated Nodal Sums - When "on", lists a calculated nodal sum for each Node ID. Off by default.

Interface Load - Lists all of the forces and moments acting on an “Interface Load” where the forces and moments are summed at a specified location in space. An “Interface Load” is defined by selected nodes and elements, using any combination of components and contributions. Components, contributions, nodes and elements are determined automatically from the selected Freebody entity (*Entity Selection* set to *From Freebody*) or selected by the user (*Entity Selection* set to *Manual Selection*).

There are 4 options when listing the *Interface Load*:

List Contribution Details - When “on”, lists each selected contribution at each Node ID. On by default.

Calculated Nodal Sums - Only available when *List Contribution Details* is turned “on”. When “on”, lists a calculated nodal sum for each Node ID. Off by default.

Calculation Details - Only available when *List Contribution Details* is turned “on”. When “on”, all details of the “Interface Load” calculation will be listed, including 9 additional columns: *d1*, *d2*, *d3*, *Fx’*, *Fy’*, *Fz’*, *Mx’*, *My’*, and *Mz’*. Off by default.

List Interface Load - When “on”, lists the ***TOTAL SUMMATION*** at the end of the listing for each Freebody ID/Output Set combination. On by default.

Destination

Messages Window - Listing is sent to the *Messages* dockable pane. The *Messages* pane must be open for this option to be available. To make is visible, use the *Tools, Other Windows, Messages* command.

Note: When listing using Interface Load with the *Calculation Details* option selected, each contribution will be listed on two lines instead of a single line. The first line contains

Data Table - Listing is sent to the *Data Table* dockable pane. The *Data Table* pane must be open for this option to be available. To make is visible, use the *Tools, Data Table* command. If the *Data Table* is “locked”, you will be prompted to “unlock”, before continuing. Each value will reside in an individual “cell”.

Clipboard - Listing is sent directly to the *Windows Clipboard* using the same format sent to the *Data Table*.

Entity Selection

Unless only one Output Set exists in the model, you will always be prompted to select Output Sets.

From Freebody - When selected, you can choose any number of existing Freebody entities from the list. Simply check the Freebody entities in the list.

For *Force Balance*, the contributions to list will match those selected in the *Freebody Contributions From* section for each selected Freebody. The nodes to list are all nodes connected to the selected *Freebody Elements*. The values will be listing in the *Coordinate System* specified for the Freebody. If the *Use Nodal Output CSys* option is selected, all listing will occur in each node’s output coordinate system.

For *Interface Load*, the contributions and coordinate system are determined in the same manner as *Force Balance*. The components to include in the summation are those selected in the *Summed Components* section, while the nodes to list are the selected *Freebody Nodes* in conjunction with the selected *Freebody Elements*.

It is possible to highlight any number of Freebody entities from the list. Holding down the “Ctrl” key will enable highlighting of multiple Freebody Entities. Holding down “Shift” while picking a “first”, then a “last” Freebody entity will highlight a range of entities. Once highlighting is complete, use the *Toggle Selected On* or *Toggle Selected Off* icon buttons to check or uncheck the highlighted entities.

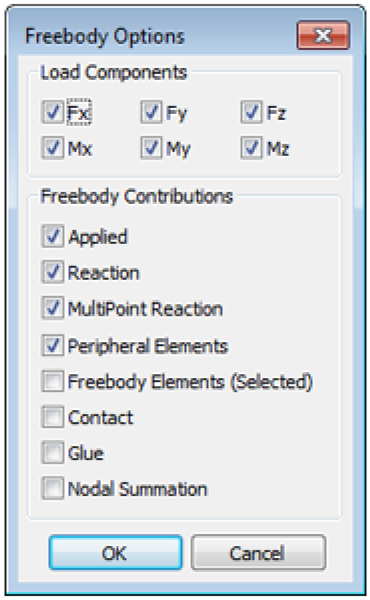
To reduce the number of entities in the list, simply enter text into the field to the left of the *Filter* icon button, then click the icon button. The list will be reduced to only those entries that contain the text you specified. You can now enter additional text, and press the *Filter* icon button again to further reduce the list. Press *Clear Filters* icon button to return to the full list and start again.

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

Note: When *Data* is set to *Interface Load* in the *List Force Balance* dialog box, only Freebody entities with *Display Mode* set to *Interface Load* or *Section Cut* will be available for selection.

Manual Selection - When selected, a number of dialog boxes will be displayed allowing you to manually specify options and select various entity types. The *Freebody Options* dialog box will be displayed first, then standard entity selection dialog boxes for Nodes, Elements, Coordinate Systems, and Output Sets will be displayed. When selected entities for *Interface Load*, you will also be prompted to define a summation location before selecting the Coordinate System.

The *Freebody Options* dialog box is only used when using *Manual Selection*



For *Force Balance*, it allows you to select only *Freebody Contributions*, while both *Load Components* and *Freebody Contributions* can be specified for *Interface Load*.

Note: The options selected in the *List Force Balance* dialog box will “persist” until changed by using the *List, Output, Force Balance* command again, using either of the icon buttons for listing in the *Freebody* tool, or until FEMAP is shut down, when it will then revert to the default settings.

For Example, this is a *Force Balance* listing using nodes and contributions selected by *Manual Selection*, for a single Output Set, using the *List Calculated Nodal Sums* option, sent to the *Message Window* (*Messages* pane):

Messages

GRID POINT FORCE BALANCE - CSys 0
OUTPUT SET: 1

Node ID	Source	Fx	Fy	Fz	Mx	My	Mz
1	F-OF-SPC	-6.30953264	-1.17596924	0.43676445	-0.24605848	0.75224721	-4.20422E-5
1	**SUM**	-6.30953264	-1.17596924	0.43676445	-0.24605848	0.75224721	-4.20422E-5
2	F-OF-MPC	5.93124533	4.1274953E-4	1.24723828	0.044652447	-1.07097685	-4.230729E-5
2	**SUM**	5.93124533	4.1274953E-4	1.24723828	0.044652447	-1.07097685	-4.230729E-5
3	F-OF-MPC	-5.93124533	-4.127495E-4	-1.24723828	-0.044652447	0.65523076	0.0001798905
3	**SUM**	-5.93124533	-4.127495E-4	-1.24723828	-0.044652447	0.65523076	0.0001798905
4	APP-LOAD	5.	0.	1.	0.	0.	0.
4	**SUM**	5.	0.	1.	0.	0.	0.
5	F-OF-SPC	-8.69046688	0.19035438	-3.43676448	-7.195502E-3	2.24775267	-2.365592E-5
5	ELEM 4 (P)	3.84836173	0.60695642	1.69097912	0.26819414	-1.04112637	1.1827961E-5
5	**SUM**	-4.84210491	0.79731083	-1.74578536	0.26099864	1.2066263	-1.182796E-5
6	ELEM 5 (P)	9.06875515	0.39579937	1.75276172	0.2083281	-0.92902321	-1.364798E-5
6	ELEM 4 (P)	-3.84836173	-0.017553698	-1.69097912	0.29341346	0.47746673	-1.38032E-5
6	**SUM**	5.22039318	0.37824568	0.061782598	0.50174153	-0.45155647	-2.745119E-5
7	ELEM 5 (P)	-9.06875515	-0.23703031	-1.75276172	0.13719143	0.34476915	5.2797819E-5
7	**SUM**	-9.06875515	-0.23703031	-1.75276172	0.13719143	0.34476915	5.2797819E-5
8	APP-LOAD	10.	0.	2.	0.	0.	0.
8	**SUM**	10.	0.	2.	0.	0.	0.
11	ELEM 5 (P)	-9.06875515	0.23703031	-1.75276172	-0.13719143	0.34476915	-5.279782E-5
11	ELEM 9 (P)	4.52157402	-0.930022	0.99559122	0.032551896	-0.2223707	6.2863583E-5
11	**SUM**	-4.54718113	-0.69299167	-0.7571705	-0.10463953	0.12239845	1.0065763E-5
12	APP-LOAD	10.	0.	2.	0.	0.	0.
12	ELEM 9 (P)	-5.45281887	0.93043476	-1.24282944	-0.07031218	0.1299917	2.7409689E-5
12	**SUM**	4.54718113	0.93043476	0.75717056	-0.07031218	0.1299917	2.7409689E-5

This is an *Interface Load* listing of a single *Freebody*, for a single output set, sent to the *Data Table*:

Data Table

Freebody ID / Output Set ID / Node ID

Freebody ID	Node ID	Output Set ID	CSys ID	Source	Fx	Fy	Fz	Mx	My	Mz
Freebody ID: 1										
Output Set ID: 1										
Node ID: 0										
1	0	1	0	**TOTAL SUMMATION**	0.	5.960464E-8	5.960464E-8	-1.192093E-7	-1.788139E-7	2.38419E-7
Node ID: 1										
1	1	1	0	F-OF-SPC	-6.309533	-1.175969	0.436764	-0.246058	0.752247	-0.0000420422
Node ID: 2										
1	2	1	0	F-OF-MPC	5.931245	0.00041275	1.247238	0.0446524	-1.070977	-0.0000423073
Node ID: 3										
1	3	1	0	F-OF-MPC	-5.931245	-0.00041275	-1.247238	-0.0446524	0.655231	0.00017989
Node ID: 4										
1	4	1	0	APP-LOAD	5.	0.	1.	0.	0.	0.
Node ID: 5										
1	5	1	0	F-OF-SPC	-8.690467	0.190354	-3.436764	-0.0071955	2.247753	-0.0000236559
1	5	1	0	ELEM 4 (P)	3.848362	0.606956	1.690979	0.268194	-1.041126	0.000011828
Node ID: 6										
1	6	1	0	ELEM 5 (P)	9.068755	0.395799	1.752762	0.208328	-0.929023	-0.000013648
1	6	1	0	ELEM 4 (P)	-3.848362	-0.0175537	-1.690979	0.293413	0.477467	-0.0000138032
Node ID: 7										
1	7	1	0	ELEM 5 (P)	-9.068755	-0.23703	-1.752762	0.137191	0.344769	0.0000527978
Node ID: 8										
1	8	1	0	APP-LOAD	10.	0.	2.	0.	0.	0.
Node ID: 11										
1	11	1	0	ELEM 5 (P)	-9.068755	0.23703	-1.752762	-0.137191	0.344769	-0.0000527978
1	11	1	0	ELEM 9 (P)	4.521574	-0.930022	0.995591	0.0325519	-0.222371	0.0000628636
Node ID: 12										
1	12	1	0	APP-LOAD	10.	0.	2.	0.	0.	0.
1	12	1	0	ELEM 9 (P)	-5.452819	0.930435	-1.242829	-0.0703122	0.129992	0.0000274097

Groups and Layers

- Added *Group, Operations, Generate Visible* command.

Automatically creates a group based on the entities currently visible in the active view. The command is "what you see is what you get", so if an entity is not currently visible, it will not be added to the new group.

When determining visibility, several aspects are taken into consideration:

First, overall visibility of an entity type, which can be fully controlled using the *Entity/Label* tab of the *View, Visibility* command or partially controlled using the Entity Display toolbar.

Second, individual entity visibility for certain element types, which is controlled using the various "entity" tabs in the *View, Visibility* command, the "visibility check boxes" in the *Model Info* tree, or the *Visibility* portion of various context-sensitive menus when using the *Select* toolbar.

Third, is the entire model being displayed or are Groups and/or Layers in use. Controlled by the *Group* tab and the *Layer* tab of the *View, Visibility* command or the *Group* and *Layer* branches in the *Model Info* tree.

Fourth, the Draw/Erase toolbar. This toolbar can be used to "erase" a portion of the model at any time.

When elements are visible, you will be asked "Ok to Include Properties and Materials Referenced by Elements in Group?". If you answer No, then only the visible elements and other visible entities will be added to the new group. If you answer *Yes*, the properties and materials referenced by the visible elements will be also be added to group.

Note: The entity types specifically designed for Aeroelastic analysis: Aero Panels/Bodies, Aero Properties, Areo Splines, or Aero Control Surfaces, cannot be placed into a group, therefore they will not be included in the new group, even if they are currently visible. This is also true for Freebody entities.

- Added *Group, Operations, Generate Elem Shape* command.

Automatically creates a single group or a number of separate groups in your model based on element shape. You simply select the elements you wish to consider, then FEMAP will prompt you with a question: "Ok to Make a Group for each Selected Element (No=One Group for All)?". Answering *Yes* will create separate groups containing all elements which have the same shape as at least one of the selected elements, while answering *No* will create one group for all elements which have the same shape(s) as the selected elements.

- Added *Group, Region, on Solid* command, which will add any regions defined using geometry on selected *Solid* or mesh associated with selected *Solid* to the active group.
- Added *Group, Coord Sys, on Element* command, which will add any coordinate system referenced by a selected element to the active group.
- Added ability to add Loads and Constraints on entity types which do not currently exist in the model to the active group. For instance, you could use *Group, Load, Elemental* to add Loads on element IDs before any elements exist in the model. Previously, these commands were unavailable until the entity type exists in the model.

Views

- Added ability to specify a "Label Group" to the *Group* tab of the *View, Visibility* command.

The *Label* section can be used to selectively label entities in your model by using *Label Active Group* or by using *Label Single Group* and selecting a group. The entity labels for a particular entity type must be turned on for labels of that entity type to be limited by the "Label Group", otherwise, no labels will be displayed for that entity type.

- Added and modified several commands on the *View, Rotate...* menu. These commands are also available on the *View and Rotation Center* icon menu on the *View* and *View-Simple* Toolbars. They may also be accessed using the *Rotate View* commands on the Quick Access Menu (right-mouse click in the Graphics window).

View, Rotate, Rotate About View Center

This is the default "Rotate Around" mode in FEMAP. This menu item simply highlights when chosen and instructs FEMAP to rotate around the current "View Center" location in the "Active View". The "View Center" can be set using the *View, Center* command (See Section 6.2.7, "View, Center..."). The "View Center" is also reset to the center of the graphics window anytime a *View, Autoscale...* command is used (See Section 6.2.3.1, "View, Autoscale, All", Section 6.2.3.2, "View, Autoscale, Regenerate All", and Section 6.2.3.3, "View, Autoscale, Visible").

The overall behavior of dynamic rotation is controlled by the *View, Rotate, Advanced Rotate...* command (Section 6.2.1.11, "View, Rotate, Advanced Rotate..."). Using this command sets *Rotation Center* to *View Center* in the *Advanced Rotation Options* dialog box.

The "Rotate About View Center" mode can be also be accessed using *View, Rotate, Dynamic...* (Section 6.2.1.2, "View, Rotate, Dynamic..."). You can use the *Options* button in the *Dynamic Display* dialog box to select this mode. Also, you are either in this mode or "Rotate About Rotation Center" mode, never both at the same time.

View, Rotate, Rotate About Rotation Center...

This menu item allows you to specify a location in space, using the standard coordinate definition dialog box, to use as the "Rotation Center". This does not change the location of the model on the screen like *View, Center* does; it simply allows rotation around another location.

This menu item also highlights when the “Rotate About Rotation Center” mode is on. When it is highlighted, all dynamic rotations will be about the center of rotation that you have specified. It will be no longer be highlighted when the “Rotate Around” mode in FEMAP is changed to *Rotate About View Center*.

The overall behavior of dynamic rotation is controlled by the *View, Rotate, Advanced Rotate...* command (Section 6.2.1.11, “View, Rotate, Advanced Rotate...”). Using this command sets *Rotation Center* to *Location* in the *Advanced Rotation Options* dialog box and the specified rotation center values will populate the *Define Location to Rotate Around* dialog box if you click the *Location...* button.

The “Rotate About Rotation Center” command can be also be accessed using *View, Rotate, Dynamic...* (Section 6.2.1.2, “View, Rotate, Dynamic...”). You can use the *Options* button in the *Dynamic Display* dialog box to select this mode and specify a rotation center. Also, you are either in this mode or “Rotate About View Center” mode, never both at the same time.

View, Rotate, Rotate Around View Axes

This menu item simply highlights when selected and informs you that rotation is occurring around the View Axis (screen axes), which is the default mode.

The overall behavior of dynamic rotation is controlled by the *View, Rotate, Advanced Rotate...* command (Section 6.2.1.11, “View, Rotate, Advanced Rotate...”). Using this command sets *Rotation Axes* to *View Axes* in the *Advanced Rotation Options* dialog box.

The “Rotate Around View Axes” command can be also be accessed using *View, Rotate, Dynamic...* (Section 6.2.1.2, “View, Rotate, Dynamic...”). You can use the *Options* button in the *Dynamic Display* dialog box to select this mode.At any time, you can be rotating around the *View Axes*, the *Model Axes*, a selected *Coordinate System*, or a specified *Vector*.

View, Rotate, Rotate Around Model Axes

This menu item simply highlights when selected and informs you that rotation is occurring around the *Model Axes*. Using *Model Axes* rotates around the “Global model axes”. Essentially, if you click and drag the cursor vertically in graphics window, the model will rotate about the Global X Axis, while clicking and dragging the cursor horizontally rotates the model around the Global Y Axis. To rotate about the Global Z Axis, hold down the *Alt* key and click and drag the cursor in any direction.

The overall behavior of dynamic rotation is controlled by the *View, Rotate, Advanced Rotate...* command (Section 6.2.1.11, “View, Rotate, Advanced Rotate...”). Using this command sets *Rotation Axes* to *Model Axes* and enables the *Single Axis Rotation* option in the *Advanced Rotation Options* dialog box.

The “Rotate Around Model Axes” command can also be accessed from the *Options* menu on the *Dynamic Rotations* dialog box (Section 6.2.1.2, “View, Rotate, Dynamic...”). At any time, you can be rotating around the *View Axes*, the *Model Axes*, a selected *Coordinate System*, or a specified *Vector*.

View, Rotate, Rotate Around Coordinate System...

This menu item allows you to select a coordinate system to rotate about using the *Select Coordinate System to Rotate About* dialog box. Once a coordinate system is selected, the menu item will highlight to inform you that rotation is occurring around a Coordinate System. Rotating around a coordinate system is very similar to rotating around the *Model Axis*, only the origin of the coordinate system is automatically used as the “rotation center”.

The overall behavior of dynamic rotation is controlled by the *View, Rotate, Advanced Rotate...* command (Section 6.2.1.11, “View, Rotate, Advanced Rotate...”). Using this command sets several options in the *Advanced Rotation Options* dialog box: *Rotation Axes* is set to *Coordinate System* and the selected coordinate system will appear in the drop-down; the *Single Axis Rotation* option will be enabled; and *Rotation Center* is set to *Coordinate System*.

The “Rotate Around Coordinate System” command can be also be accessed using *View, Rotate, Dynamic...* (Section 6.2.1.2, “View, Rotate, Dynamic...”). Use the *Options* button in the *Dynamic Display* dialog box to select this mode and select a coordinate system. At any time, you can be rotating around the *View Axes*, the *Model Axes*, a selected *Coordinate System*, or a specified *Vector*.

View, Rotate, Rotate Around Vector...

This menu allows you to specify a vector that will be used as the rotation axis for dynamic rotation. When you specify a rotation vector, the base of the vector is automatically used as the “rotation center”, so all rotations will be about that point. Once a vector is specified, this menu item will highlight to inform you that rotation is occurring around a specified Vector.

The overall behavior of dynamic rotation is controlled by the *View, Rotate, Advanced Rotate...* command (Section 6.2.1.11, “View, Rotate, Advanced Rotate...”). Using this command sets several options in the *Advanced Rotation Options* dialog box: *Rotation Axes* is set to *Vector* and the specified vector values will populate the *Define Vector to Rotate Around* dialog box if you click the *Vector* button, the *Rotation Center* is set to *Location*, and the values of the *Base* coordinates of the specified vector will populate the *Define Location to Rotate Around* dialog box if you click the *Location...* button.

Note: This command does not account for any perspective that you have specified. It is usually best therefore to turn off *Perspective* before using this command.

The “Rotate Around Vector” command can also be accessing using the *View, Rotate, Dynamic...* (Section 6.2.1.2, “View, Rotate, Dynamic...”). Use the *Options* button in the *Dynamic Display* dialog box to select this mode and specify a vector. At any time, you can be rotating around the *View Axes*, the *Model Axes*, a selected *Coordinate System*, or a specified *Vector*.

View, Rotate, Roll-Thru...

This menu item allows you to enter a unique display mode which essentially places you “inside the model” and allows you to “look around” as you rotate the model. To allow you to see “inside” the model, the model will be “clipped” using the screen as the “clipping plane”, with center of rotation being at the “screen center”. As the model is rotated, portions of the model will disappear while other portions appear. Once this mode is selected, this menu item will highlight to inform you are in “Roll-Thru” mode.

The overall behavior of dynamic rotation is controlled by the *View, Rotate, Advanced Rotate...* command (Section 6.2.1.11, “View, Rotate, Advanced Rotate...”). Using this command sets several options in the *Advanced Rotation Options* dialog box: *Rotation Axes* is set to *View Axes*, the *Model Always in Front of Viewer* option is disabled, and the *Rotation Center* is set to *Screen Center.*

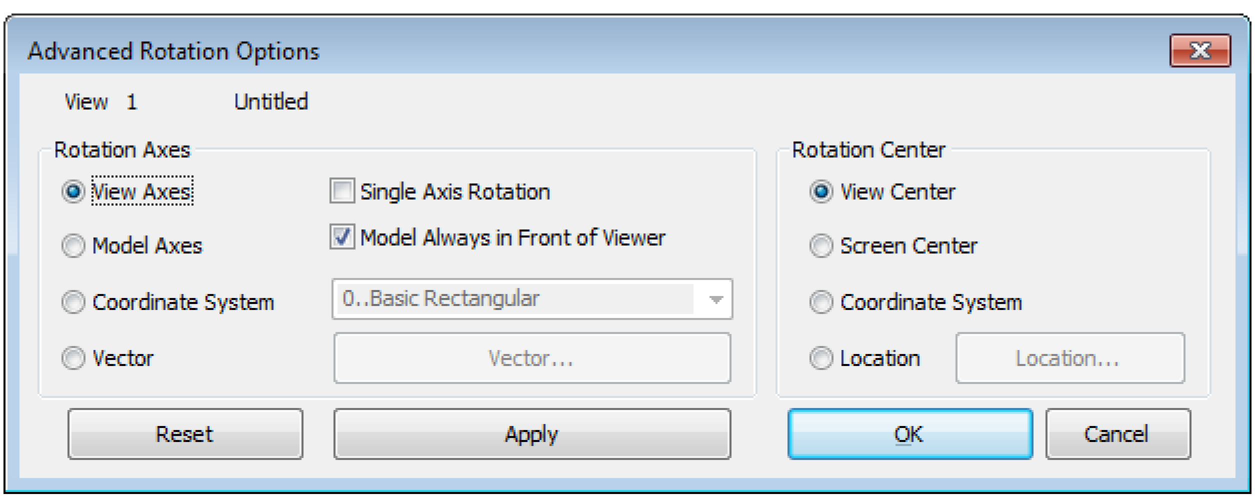
Note: The “Roll-Thru” functionality is designed to work appropriately only when the above settings are specified in the *Advanced Rotation Options* dialog box. If the settings for *Rotation Axes* or *Rotation Center* are changed, the model will still rotate and portions of the model will be still be “clipped”, but it may not behave as expected and the *Roll-Thru* menu item will no longer be highlighted.

The “Roll-Thru” command can also be accessing using the *View, Rotate, Dynamic...* (Section 6.2.1.2, “View, Rotate, Dynamic...”). Use the *Options* button in the *Dynamic Display* dialog box to select this mode.

Note: At any time, you can dynamically update the location where clipping is occurring by holding down the *Shift* and *Alt* keys at the same time, then scrolling the mouse wheel or “clicking and dragging” with the middle mouse button or mouse wheel. If you are not in a command, this same keyboard combination will also work to update the location when “clicking and dragging” with the left mouse button.

View, Rotate, Advanced Rotate...

This menu item controls the overall behavior of dynamic rotation. Many of the commands on the *View, Rotate...* menu set one or more options in the *Advanced Rotation Options* dialog box, which essentially changes how dynamic rotation behaves in varying degrees.



Note: The commands on the *View, Rotate...* menu represent the most commonly used combinations of *Rotation Axes* options and *Rotation Center*, therefore you should probably only use this command if you want to have dynamic rotation behave in uncommon manner. For instance, it may be beneficial for you to have *Rotation Axes* set to *Coordinate System*, but want the *Rotation Center* set to *Location*, which is not possible to achieve without using the *Advanced Rotation Options* dialog box

There *Rotation Axes* section controls the axes or axis which dynamic rotation will occur about in the graphics window. The default value is *View Axes*, which is essentially the screen axes. Using *Model Axes* rotates around the "Global model axes". Essentially, if you click and drag the cursor vertically in graphics window, the model will rotate about the Global X Axis, while clicking and dragging the cursor horizontally rotates the model around the Global Y Axis. To rotate about the Global Z Axis, hold down the *Alt* key and click and drag the cursor in any direction. *Coordinate System* works similar to *Model Axes*, but rotates around the axes of the coordinate system selected in the drop-down instead of the "Global model axes". Both *Model Axes* and *Coordinate System* behave best when the *Single Axis Rotation* option is also enabled (see Section 6.2.1.12, "View, Rotate, Single Axis Rotation"). *Vector* allows you to rotate about a vector, which you specify using the *Vector* button, then clicking and dragging the cursor in any direction simply rotates around the vector.

The *Model Always in Front of Viewer* option is enabled by default and should always be enabled unless you are using the "Roll-Thru" functionality (see Section 6.2.1.10, "View, Rotate, Roll-Thru..."). When this option is disabled, you are able to move "inside" the model where you can then "look around inside", but this only works as expected when *Rotation Axes* is set to *View Axes* and *Rotation Center* is set to *Screen Center*.

The *Rotation Center* section controls the "center of dynamic rotation". The default value is *View Center*, which is essentially the center of active view and include "depth into the screen" to account for the overall size of the model. The *Screen Center* is similar to the *View Center*, only no "depth into the screen" is considered. The option should only be selected when using the "Roll-Thru" functionality as it may cause unexpected behavior otherwise. *Coordinate System* will use the origin of the coordinate system selected in the drop-down of the *Rotation Axes* section as the "rotation center". The *Location* option allows you to specify a location in space to use as the rotation center. Use the *Location...* button to use the standard coordinate definition dialog box to specify the location.

At any time, you can click the *Reset* button, which sets *Rotation Axes* to *View Axes*, enables the *Model Always in Front of Viewer* option, and sets *Rotation Center* to *View Center*.

View, Rotate, Single Axis Rotation

This menu item is simply a "toggle" used to enable or disable "Single Axis Rotation" and controls how model XY rotations will be done. Typically, dragging horizontally rotates around Y, dragging vertically rotates around Z. Normally, it is easiest to accomplish compound rotations by dragging along one axis for a while, then dragging along the other. Since it is very difficult to drag the mouse along a precisely horizontal or vertical axis however, the *Single Axis Rotation* option limits rotations to the single direction in which you are moving the greatest distance. Small deviations from horizontal or vertical will be ignored. If you turn this option off however, moving the mouse diagonally will perform a rotation about a diagonal vector. When in use, menu item will be highlighted and "checked".

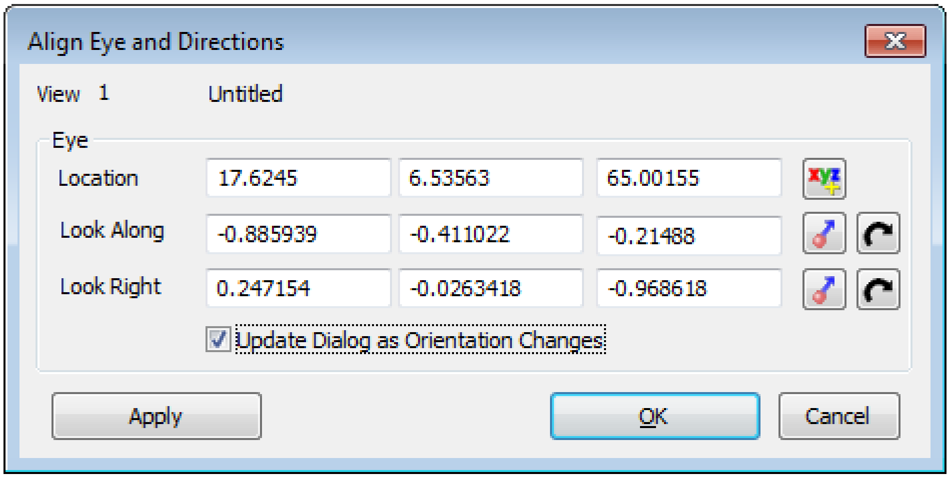
The overall behavior of dynamic rotation is controlled by the *View, Rotate, Advanced Rotate...* command (Section 6.2.1.11, "View, Rotate, Advanced Rotate..."). Using this command simply enables the *Single Axis Rotation* option in the *Advanced Rotation Options* dialog box.

The "Single Axis Rotation" command can be also be accessed using *View, Rotate, Dynamic...* (Section 6.2.1.2, "View, Rotate, Dynamic..."). Use the *Options* button in the *Dynamic Display* dialog box to select this mode.

- Added *View, Align By, Eye and Directions* command.

View, Align By, Eye and Directions...

... allows you to align the view by entering a *Location* for your "eye", a *Look Along* direction, and a *Look Right* direction. The *Look Along* direction is specified by selecting a location in space that represents you looking straight ahead from the specified "eye" *Location*, while the *Look Right* direction is specified by selected a location in space that represents you looking to your right



You can use the *Locate Eye* icon button to use the standard coordinate definition dialog box to select the "eye" *Location*. You can use the *Define Vector* icon button next to the *Look Along* fields to specify a vector for the *Look Along* direction where the default values for *Base* in the standard vector definition dialog box are the current "eye" *Location* coordinates. The *Rotate Around* icon button for *Look Along* allows you to enter an angle to "rotate around" the vector from the "eye" *Location* to the *Look Right* location. Similar icon buttons exist for Look Right and can be used to enter a vector to define the Look Right location or "rotate around" the vector from the "eye" *Location* to the *Look Along* location.

When the *Update Dialog as Orientation Changes* option is enabled, the values will reflect any orientation change you make with the mouse and middle mouse button, Spaceball, touch interface, etc.

- Added the ability to control visibility of individual elements. This can be done using the Visibility menu on the context-sensitive menu for elements when Element is the active entity type in the Select Toolbar or by using the Hide Individual Elements or Show All Individual Elements commands on the "Visibility check box" context-sensitive menu for *Elements, By Type* and *Elements, By Shape* in the *Model Info* tree.
- Added ability to double-click in the mouse wheel or middle mouse button to run the *View, Autoscale, Visible* command, when not in another command. Holding down *Shift* and double-clicking the mouse wheel or middle mouse button when not in another command will run the *View, Autoscale, All* command.
- Added ability to display the name of the element quality check currently specified in the *Surface Mesh Quality* tool of the *Meshing Toolbox* as a *Post Title*, when the Surface Mesh Quality is being displayed.
- Updated the *View Legend* to have a ":" between the item in the legend and the ID(s). Also, the *Label Group* has been added the *View Legend* and is shown as "LG:(Group ID)". Finally, items in the *View Legend* and the *Post Titles* can now have up to 2 characters to the left of the ":".
- Updated visibility of loads and constraints to not display them when the entity on which they are applied is also not currently visible. This behaves in this manner for mesh-based and geometry-based loads and constraint. Also, if all nodes of a constraint equation are not visible, the constraint equation will also not be displayed.

Output and Post-Processing

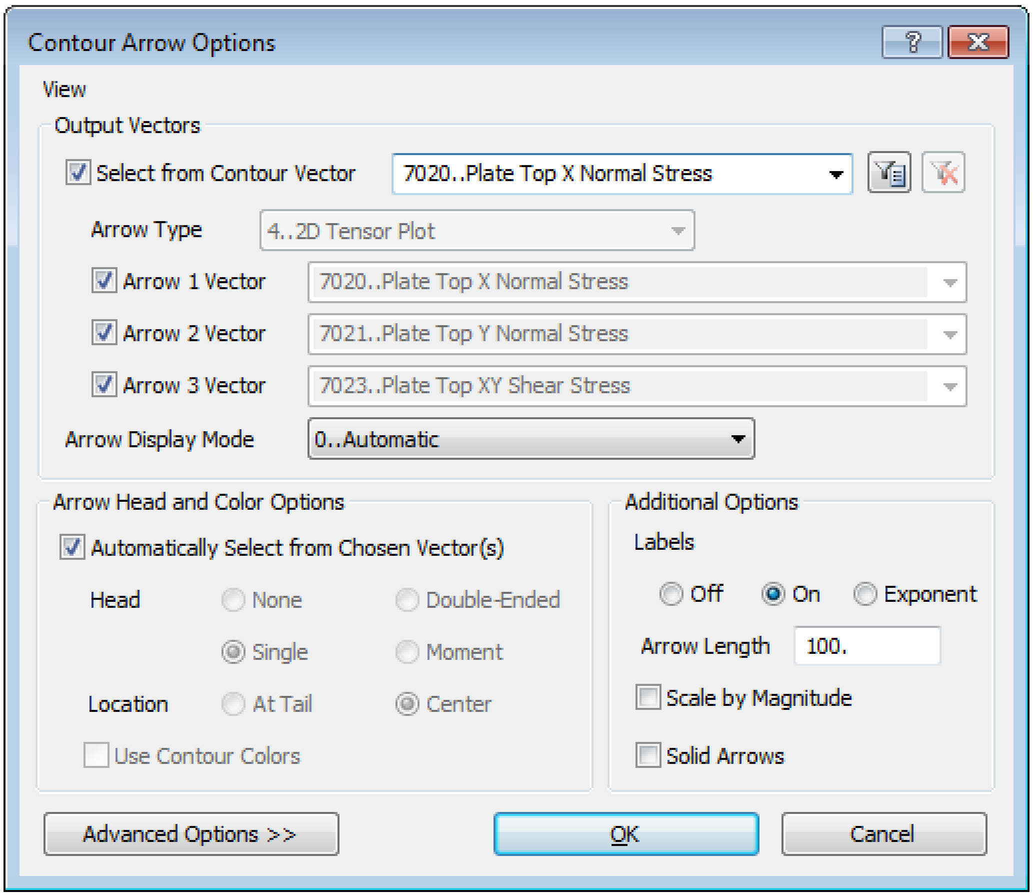
- Added ability to include Max/Min information from the currently displayed output vector in the *Post Titles.* To display the Max/Min information, use the *View, Options* command, set *Category* to *PostProcessing*, select *Post Titles* from the *Options* list, then set *Legend Style* to either "3..Titles and Min/Max Data" or "4..Titles and Min/Max/Average".
- Added option to display the colors in the *Contour/Criteria Legend* in reverse order. To display the colors in reverse order, use the *View, Options* command, set *Category* to *PostProcessing*, select *Contour/Criteria Legend* from the *Options* list, then set *Position* to "8..Top Left, Reversed", "9..Top Center, Reversed", "10..Top Right, Reversed", "11..Center Left, Reversed", "12..Center Right, Reversed", "13..Bottom Left, Reversed", "14..Bottom Center, Reversed", "15..Bottom Right, Reversed".

- Updated *Model, Output, From Load* command to automatically support creation of varying output when selecting Elemental Pressures that vary at the face corners.
- Updated the name of the *Contour Style* option *Vector* to *Arrow*. Also, added options to automatically select the type of arrow plot, arrow style, any additional contour vectors, and other options based on the output vector currently specified in the *Contour* drop-down of the *Select PostProcessing Data* dialog box.

The *Contour Arrow Options* dialog box is used to control the display of contour results as an Arrow Plot.

By default, creating a plot of contour arrows is as simple as creating a contour plot. You simply specify an output vector in the *Contour* drop-down in the *Select PostProcessing Data* dialog box and that output vector will then be displayed as an arrow plot. The type of arrow plot displayed, any additional output vector(s) presented with the selected output vector, and a number of *Arrow Head and Color Options* are automatically determined by the program using predetermined criteria.

In many cases, it may not be necessary to set any additional options, but if the "default" arrow plot is not the desired plot, the plot can be customized using settings in the *Contour Arrow Options* dialog box:



Output Vectors

This section of the dialog box is essentially managed by the *Select from Contour Vector* option, as most options are not available when it is enabled, however, the *Arrow Display Mode* and the visibility of individual arrows can be modified at all times. In addition, the *Filter Output Vectors* and *Clear Filter* icon button can always be used to reduce the number of output vectors available for selection in any drop-down used for output vector selection.

Note: For more information on using the *Filter Output Vectors* functionality, see "Making it Easier to Select Output Data - Using the Output Vector Filters" in Section 8.2.2.2, "Selecting Data for a Deformed or Contour Style".

Select from Contour Vector - When this option is enabled, which is the default, simply select the "seed" output vector using the drop-down next to this option, which will automatically select the *Arrow Type* and the output vectors selected in the *Arrow 1 Vector*, *Arrow 2 Vector*, and *Arrow 3 Vector* drop-downs. If disabled, all of the previously unavailable options become available for modification and the "seed" output vector becomes unavailable.

Arrow Type - When available, there are 5 available options: *Standard Output Vectors*, *Single Value*, *2D Components*, *3D Components*, and *2D Tensor Plot*.

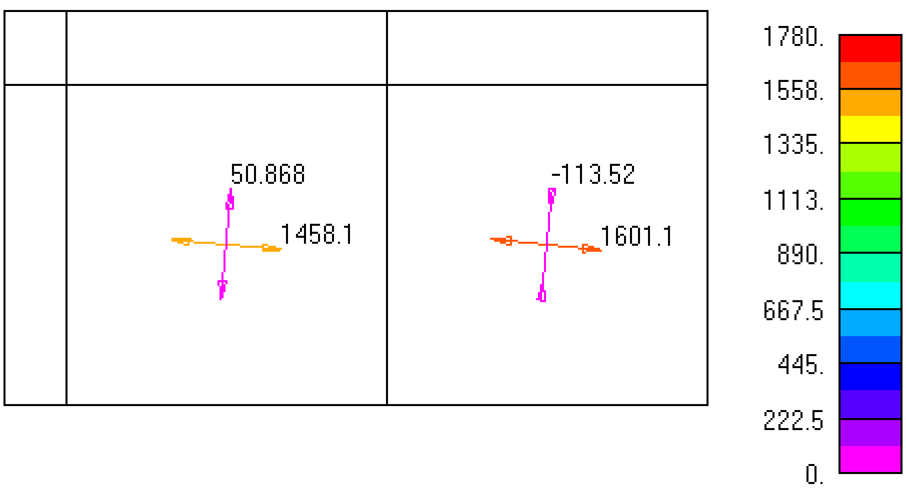
Note: In many cases, Arrow plots can be displayed using transformed output, which can be quite useful. When shell and solid output is not transformed, it is always displayed in the "current output orientation", which can be viewed in the *Current Output Orientation* dialog box. A single element shape can have different output orientations specified for different types of shell and solid output, but this is somewhat uncommon. To access the *Current Output Orientation* dialog box, click the *Transform* button in the *Select PostProcessing Data* dialog box, then by press the *Output Orientation* button. See "Transforming Your Output" in Section 8.2.2.2, "Selecting Data for a Deformed or Contour Style".

Standard Output Vectors - Automatically calculates the direction and magnitude of each selected vectors to generate the arrow plot. You can select up to three vectors for each plot. Thus, for solid Principal stresses you could pick *Solid Major* for *Arrow 1 Vector*, *Solid Intermediate* for *Arrow 2 Vector*, and *Solid Minor* for *Arrow 3 Vector* to see vectors for all three Principal stresses on one plot.

Note: This option is never shown as *Arrow Type* when using *Select from Contour Vector*, as this functionality has been incorporated into the other *Arrow Type* options, where everything is automatically selected. For example, the solid Principal stresses will be displayed with *Arrow Type* set to *3D Components* and rotated using the appropriate Principle stress angle.

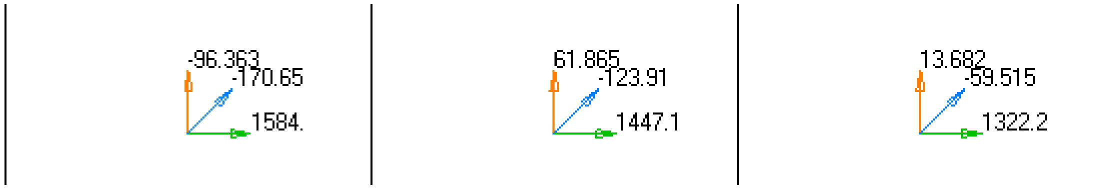
Single Value - Simply displays the output vector selected using the *Arrow 1 Vector* drop-down. You should also define a display direction by using the *Vector* button next to the *Arrow Type* drop-down, which allows you to select a vector using the standard *Vector Definition* dialog box. This *Arrow Type* is useful when plotting invariants, such as *Von Mises Stress*.

2D Components - Displays the output vectors selected in the *Arrow 1 Vector* and *Arrow 2 Vector* drop-downs as two different arrows. The components are displayed in the current output orientation for the element shape and the type of output selected. This *Arrow Type* is useful when plotting Principle stresses on 2D elements.



3D Components - Displays the output vectors selected in *Arrow 1 Vector*, *Arrow 2 Vector*, and *Arrow 3 Vector* drop-downs as three different arrows. The components are displayed in the current output orientation for the element shape and the type of output selected. This *Arrow Type* is useful when plotting Principle stresses on 3D elements or component nodal output, such as translations, applied forces, or constraint forces.

2D Tensor Plot - Creates a plot of three vectors on a 2D element (i.e.,"Crow's Foot" plot). The *Arrow 1 Vector* output vector will be plotted in the X-direction (based on current output orientation specified for the element shape), the *Arrow 2 Vector* output vector will be plotted in the Y-direction, and the *Arrow 3 Vector* will be plotted at a 45 degree angle between the X-direction and the Y-direction. Although it may be used to show any three vectors on 2D elements, this is very useful when showing three related vectors, with two being "directional" and one being "shear", such as *Plate X Normal Stress*, *Plate Y Normal Stress*, and *Plate XY Shear Stress*:



Arrow 1 Vector - Used to specify the output vector to display as “Arrow 1”. Check box to the left of *Arrow 1 Vector* controls visibility of “Arrow 1”. Used by all *Arrow Type* options.

Arrow 2 Vector - Used to specify the output vector to display as “Arrow 2”. Check box to the left of *Arrow 2 Vector* controls visibility of “Arrow 2”. Available for all *Arrow Type* options expect *Single Value*.

Arrow 3 Vector - Used to specify the output vector to display as “Arrow 3”. Check box to the left of *Arrow 3 Vector* controls visibility of “Arrow 3”. Available when *Arrow Type* is set to *Standard Output Vectors*, *3D Components*, and *2D Tensor Plot*

Arrow Display Mode - Allows you to select how the Arrows will be displayed. The default setting for this option is *Automatic*, which automatically selects an *Arrow Display Mode* based on the *Arrow Type* option and the selected output vector(s). *Automatic* will display all “Total Vectors” as resultants, while all others are displayed as components. Additional options include:

Components - displays individual arrows for *2D Components* (*X* and *Y*) and *3D Components* (*X*, *Y*, and *Z*).

Resultant - displays a single resultant vector for *2D Components* or *3D Components*.

Shear (XY)/Axial (Z) - displays *X* and *Y* arrows as resultant (*“Shear”*) and the Z arrow as a component (*“Axial”*).

Shear (XZ)/Axial (Y) - displays *X* and *Z* arrows as resultant (*“Shear”*) and the Y arrow as a component (*“Axial”*).

Shear (YZ)/Axial (X) - displays *Y* and *Z* arrows as resultant (*“Shear”*) and the X arrow as a component (*“Axial”*).

Arrow Head and Color Options

This section of the dialog box is fully managed by the *Automatically Select from Chosen Vector(s)* option, as none of the other options are available when it is enabled.

Automatically Select from Chosen Vector(s) - When this option is enabled, which is the default, the type of arrow head, the location of the arrows, and if arrows are displayed using “contour colors” or “arrow colors” is automatically determined by the *Arrow Type* and output vector(s) specified in the *Output Vectors* section. If disabled, all of the options in this section become available for modification.

Head - Choose a type of arrow head. Choices are *None*, *Double-Ended* (arrowheads on both ends), *Single* (arrowhead on one end), or *Moment* (two arrowheads on one end).

Location - Choose between *At Tail* (depending on type of output, Arrow extends out from node or element centroid) or *Center* (Arrow centered at node or element centroid).

Use Contour Colors - When on, Arrows are displayed using “Contour Colors” and the *Contour Legend* is also visible. When off, Arrows are displayed using individual *Arrow Color* and the *Contour Legend* is not visible.

Note: Currently, the individual “arrow color” for each arrow can only be changed by clicking the *Advanced Options>>* button in the *Contour Arrow* options dialog box, then changing the *Color* beneath the appropriate *Arrow (#)* in the *Output Vectors* section of the *Advanced Contour Arrow Colors* dialog box or by using the *Arrow Color* field under each *Arrow (#) Vector* field in the *PostProcessing Toolbox*.

Additional Options

This section of the dialog box contains options which are available at all times, which can be used to control labeling, length, and overall appearance of all contour arrows.

Labels - Choose *Off* (no arrows shown with labels), *On* (all arrows shown with labels), or *Exponent* (same as *On*, only labels are shown as exponents).

Note: Currently, the number of displayed digits for labels can only be controlled by using the *Label Digits* field in *Contour Tool* of the *PostProcessing Toolbox* or by using the *View, Options* command, choosing *PostProcessing* as the *Category,* selecting *Contour Arrow Options* from the *Options* list, then entering a value into the *Digits* field. Enter a value from 0 to 10. Entering 0 will display a default number of digits.

Arrow Length - Allows you to specify a length for all Contour Arrows (0.0 to 1,000.0, default is 100.0).

Scale by Magnitude - When on, Arrows are displayed based on magnitude, with the Maximum value displayed using 1.0**Arrow Length* and Minimum value displayed using *Min Length* (0.25 default)**Arrow Length*.

Solid Arrows - When on, Arrows are displayed using solid representations. When off, Arrows are wireframe.

Advanced Options>> button

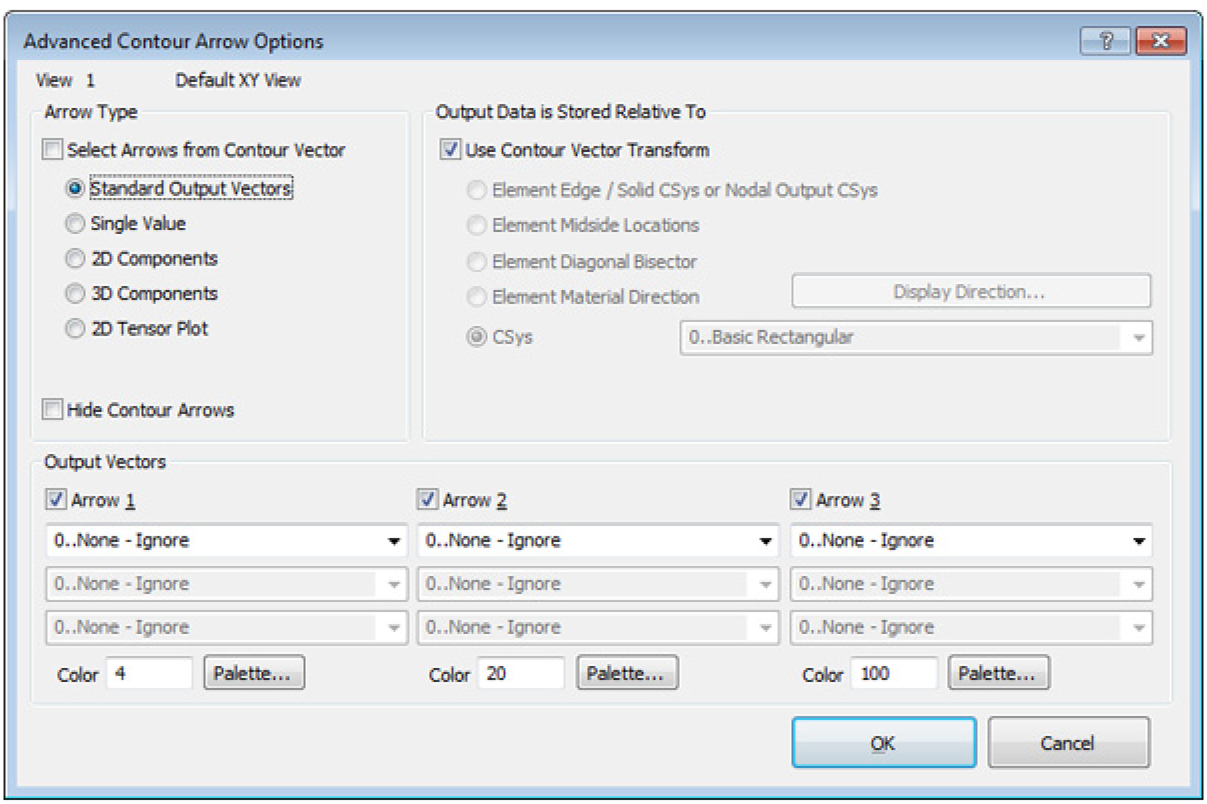
This button opens the *Advanced Contour Arrow Options* dialog box, which allows you to setup Contour Arrows using a "legacy" dialog box found in earlier versions of FEMAP.

By default, the *Select Arrows from Contour Vector* option will be enabled and the only items available are the *Color* fields and *Palette* buttons in the *Output Vectors* section, along with the *Display Direction* button in *Output Data is Stored Relative To* section, but this is only available when *Arrow Type* is set to *Single Value*.

Note: The *Advanced Contour Arrow Options* dialog box should ONLY be used if all possibilities in the *Contour Arrow Options* dialog box have been explored and the desired arrow plot cannot be generated. One case where this dialog box may be needed would be to create an arrow plot of three independent output vectors in three different directions, but this is somewhat atypical.

In addition, there is no known reason to turn off the *Use Output Vector Transform* option in *Output Data is Stored Relative To* section of this dialog box. If this option is disabled, then the output vectors will be based on the option selected in the *Output Data is Stored Relative To* section, instead of the current output orientation, therefore, the output vectors CANNOT BE TRANSFORMED.

When the *Select Arrows from Contour Vector* option is disabled, the options in the *Arrow Type* section and the appropriate drop-downs in the *Output Vectors* section will become available for selection. Notice, the options in the *Output Data is Stored Relative To* section are still unavailable to modify, which is by design.



The *Arrow Type* section offers the same five methods as the *Contour Arrows Options* dialog box- *Standard Vectors*, *Single Value*, *2D Components*, *3D Components*, or *2D Tensor Plot*. In all cases except *Single Value*, you can display up to three contour vectors on the same arrow plot. As long as the *Use Contour Vector Transform* option is enabled in *Output Data is Stored Relative To* section, the arrow plots generated using this dialog box should be the same as ones generated in the *Contour Arrow Options* dialog box, provided all output vectors are properly selected using the appropriate drop-downs in this dialog box.

Note: The documentation for the *Advanced Contour Arrow Options* dialog box which follows is from an earlier version of FEMAP. It assumes the *Use Contour Vector Transform* option has been disabled, which is not recommended.

Remember, if this option is disabled, then the output vectors will be based on the option selected in the *Output Data is Stored Relative To* section, instead of the current output orientation, therefore, the output vectors CANNOT BE TRANSFORMED.

Typically you will want to select a standard vector since these output vectors contain data that was already calculated based upon their components. Examples of these type of vectors include Total Translation or Rotation for nodal data, Plate Top Major Stress for 2-D plate elements, and Solid Major Principal Stress for 3-D solid elements.

When *Standard Vectors* is selected, you only need to select one output for each value. The direction and magnitude of these vectors are automatically calculated by FEMAP to generate the vector contour plot. You can generate up to three vectors for each plot. Thus, for solid principal stresses you could pick *Solid Major* for Vector 1, *Solid Intermediate* for Vector 2, and *Solid Minor* for Vector 3 to see vectors for all three Principal stresses on one plot.

If you want to display vectors other than standard, you can simply select the number of components of the vector (single, 2D, or 3D). You will need to select output for that number of components for each vector displayed. If you wish to see three 3D vectors, you will need to select nine output vectors. If you decide to display a single vector, you must also define the vector direction under the *Display Direction* button. The standard FEMAP *Vector Definition* dialog box is displayed to enable you to define the direction of this 1D plot.

If you select 2D components, and the two associated vectors for the X and Y components for Vector 1 (and Vector 2 and 3 if multiple vectors are required), the data will be displayed in the coordinate directions defined by the *Output Data is Stored Relative To* portion of the dialog box (discussed below). 3D components works just like 2D, except that you must now choose three vectors for Vector 1 (and Vectors 2 and 3 if required). In either of these cases, the components must be chosen in a top-down fashion. That is, the first vector represents the X component, the second is the Y component, and the third (if necessary) is the Z component. By allowing you to select a standard vector, or any combination of 1D, 2D, or 3D vectors, FEMAP can produce elemental centroidal plots for a wide variety of conditions and vectors.

Note: FEMAP does not know, nor keep track of the coordinate system where your output is defined. Therefore it is up to you to provide this information prior to making a vector plot. You can choose any one of five different methods, depending on how your results were defined by your analysis program.
This is extremely important! If you choose the wrong or inappropriate method, the display will be wrong. You must know how the data was defined by your analysis program before proceeding.

The 2D Tensor Plot (i.e.,"Crow's Foot" plot) allows you to create a plot of three vectors on a 2D element. *Vector 1* will be plotted in the X-direction (based on the option selected in the *Output Data is Stored Relative To* section), *Vector 2* in the Y-direction, and *Vector 3* at a 45 degree angle between the X-direction and the Y-direction. Although it may be used to show any three vectors on 2D elements, this is very useful when showing three related vectors, with two being "directional" and one being "shear", such as Plate X Normal Stress for *Vector 1 (X)*, Plate Y Normal Stress for *Vector 2 (Y)*, and Plate XY Shear Stress for *Vector 3 (XY)*.

Output Data is Stored Relative To

Element Edge / Solid CSys or Nodal Output CSys - This method is used for output from planar elements if the X output direction is defined relative to the first element edge (the line connecting the first two nodes). It is used for solid elements if output is defined in the solid property coordinate system. Use it for nodal output, if the results are in the nodal output coordinate system. Be aware, however, that for most standard three dimensional nodal output vectors (displacements, constraint forces, applied loads, velocities and accelerations) FEMAP transforms output into global coordinates and this option is not appropriate.

Element Midside Locations - This option is used for output from planar elements when the X output direction is defined as the vector that joins the midsides of the second and final (4th for a quad, 3rd for a triangle) edge.

Element Diagonal Bisector - This option is used for output from planar elements when the X output direction is defined as the vector that bisects the angle formed by the two element diagonals, in the quadrant that generally points along the first edge.

Element Material Direction - This option is used for elemental output that is defined in the material direction; for example, along the rotated plies of a laminate.

CSys - This final method is available if your output is defined in some known coordinate system. You must also choose the appropriate coordinate system along with this option.

Geometry Interfaces

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

FEMAP Interface	Latest Supported Version
Parasolid	Parasolid 29.0
Solid Edge	Solid Edge with Synchronous Technology 8
Pro/Engineer	CREO 3
CATIA	CATIA V5-6R2015 SP3
ACIS	ACIS 26, SP1
SolidWorks	SolidWorks 2016

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
- Analysis Manager
- NX Nastran Interface
- Nastran Interfaces (NX and MSC/MD)
- ANSYS Interface
- ABAQUS Interface
- DYNA Interface
- PATRAN Interface
- I-DEAS Interface

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

FEMAP Neutral File Interface

- Updated Neutral Read and Write for v11.3 changes
- Updated all FEMAP Neutral file converters from version 10.0 and above to be 64.bit applications.
- Added ability to select the *Write Output* option in the *Entity Options* section of the *Neutral File Write Options* dialog box when *Group Only* is selected. This will write only output to the neutral file for entities in the group. This option is off by default and must be turned on to write the output for the specified group.
- Added Binary Output File (FNO) documentation to Neutral File docs (neutral.pdf).
- Added the “Automatic Add” group ID to the Neutral File.

Analysis Manager

- Added *Renumber* button to the *Analysis Set Manager* dialog box, which allows you to renumber either analysis sets or analysis cases in an individual analysis set. To renumber analysis cases, a subcase must be highlighted, otherwise, it will prompt you to renumber analysis sets.
- Added functionality to *Boundary Conditions* dialog box in the *Analysis Set Manager* which allows you to specify "-1..Use Active Constraint Set" for *Constraints* and/or "-1..Use Active Load Set" for *Loads*, which will use the boundary conditions specified in the active Constraint Set and/or the loads specified in the active Load Set.
- Updated all Nastran analysis manager dialog boxes to use the same *NASTRAN* title

NX Nastran Interface

- Added support to read and write PLASALG and corresponding value for the NXSTRAT entry.
- Added ability to read the analysis type from the CASECC data block when using SOL 601.
- Added support to handle Drilling Grid Point Force output.
- Added support to import and attach to output on Solid Laminate elements found in *.op2 files created by NX Nastran 11.0.
- Added support to write MAT3 entry for plane strain plane stress elements (CPLSTNi, CPLSTSi elements).
- Updated translator to not write nothing in the NORM field of the EIGC entry, as NX Nastran has removed the option. The *Normalization Method* section in the *NASTRAN Modal Analysis* dialog box becomes unavailable when any option in the *Complex Solution Methods* section is selected.
- Updated translator for Rotor Dynamics to always write the EIGRL entry using MASS normalization no matter what the *Normalization Method* option is set to in the *NASTRAN Modal Analysis* dialog box.

A number of bugs were corrected

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

Nastran Interfaces (NX and MSC/MD)

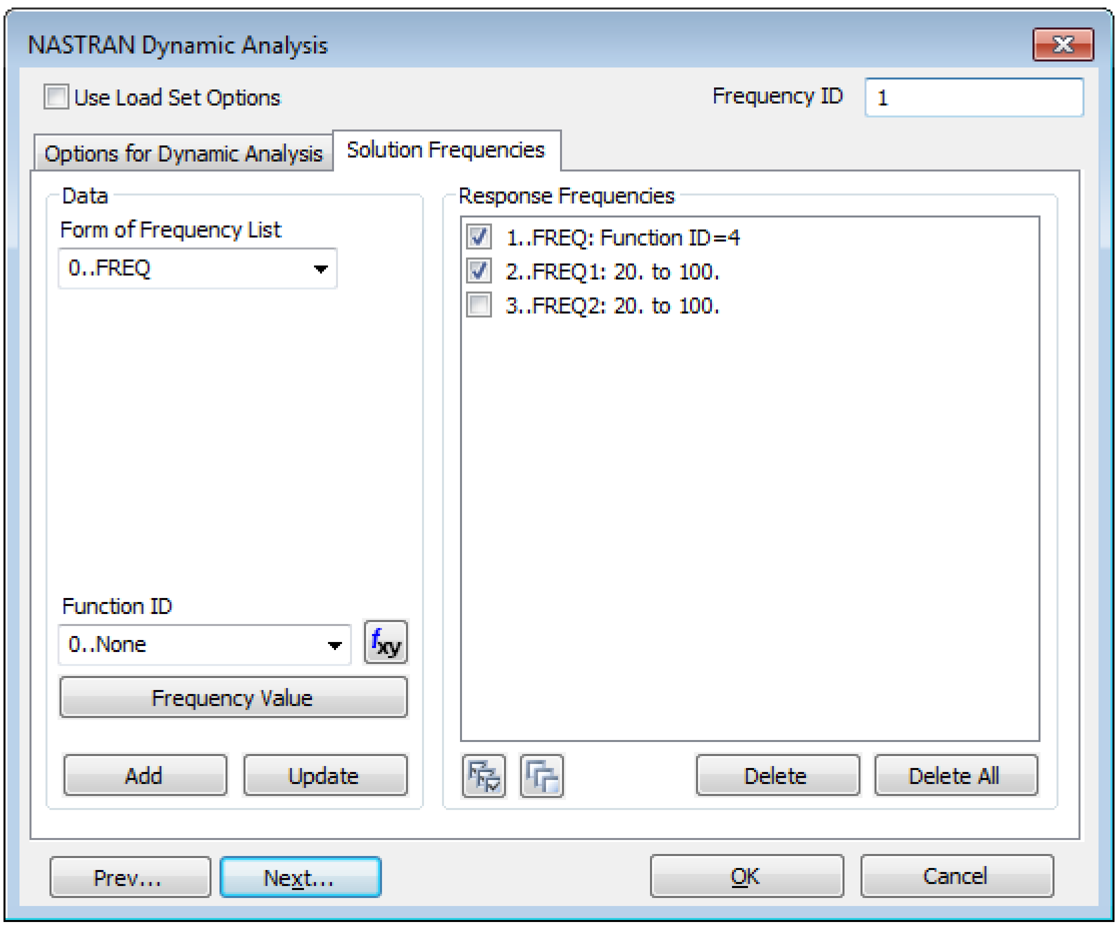
- Added *Additional Command Line Arguments* field to the *NASTRAN Executive and Solution Options* dialog box. This allows the user to include command line arguments which are not explicitly supported by FEMAP, when launching Nastran. If any command line arguments are specified in the *Arguments* field for *NX Nastran*, *MSC/MD Nastran*, and/or *Autodesk Nastran* on the *Solvers* tab of *File, Preferences*, they will also appear in the *Additional Command Line Arguments* field when a new analysis set is created using the corresponding solver.
- Added functionality which will attempt to use the Nastran Subcase ID as the FEMAP output set ID. If an output set of that ID already exists, it will use the next available output set ID higher than the Nastran Subcase ID. To enable this functionality, turn on the *Use Static Subcase IDs* option on the *Results* tab of *File, Preferences*.
- Added functionality which will add a revision number when creating a new output set which corresponds to a Nastran Subcase ID, when a FEMAP output set corresponding to that Nastran subcase ID already exists in the model. To enable this functionality, turn on the *Track Revision* option on the *Results* tab of *File, Preferences*.
- Added support to read strain output on CBEND elements from the *.op2 file.
- Added support to write NonZero Constraints as SPC entries. An individual SPC entry will be written for each non-zero value specified on a node. When reading SPC entries from a Nastran input file, a single constraint will be created if multiple SPC entries have the same SID value and G1 value.
- Added support to read and write the RANDT1 entry, which specifies values for autocorrelation function time lag. These values can be specified in the *Autocorrelation Function Time Lag* section of the *NASTRAN Power Spectral Density Factors* dialog box.
- Added support to read and write XYPRINT\PEAK velocities, which can be selected by checking the appropriate check boxes in the *Nodal Output Requests* section of the *NASTRAN Output for Random Analysis* dialog box. Also, added support to create PSDF and/or AUTO XYPRINT\PEAK entries, which can be specified by checking the *PSDF* and/or *AUTO* check boxes in the *NASTRAN Output for Random Analysis* dialog box. Finally, added reading XYPRINT\PEAK AUTO\PSDF results from the *.f06 file.

- Updated the *Varying Translational Acceleration* body load, which creates an ACCEL entry, to allow the user to define an acceleration where the acceleration direction is aligned with the direction of acceleration variation.
- Updated how the TIME executive control entry is written. When creating a new analysis set, the default value for *Max Time (in minutes)* in the *NASTRAN Executive and Solution Options* dialog box will be 0, which causes the TIME entry to not be written. TIME will only be written when the user sets the value above 0.
- Updated Nastran interface so that real values between 100,000 and 1,000,000 come out as a full 8 character wide field as long as you have the *Improve Real Number Precision* option enabled on the *Interfaces* tab of *File, Preferences*. Previously some cases came out as 7 characters and lost the first decimal digit.
- Updated the name of the *Advanced Options* tab in the *NASTRAN Dynamic Analysis* dialog box to be *Solution Frequencies* and modified how list of solution frequencies are defined to allow more than two FREQ or FREQi entries to be written to a Nastran input file or read in from a Nastran input file.

The *Solution Frequencies* tab may be used to define the "solution frequencies" in an alternate way for direct frequency response, modal frequency response, and random response analysis.

This tab offers two sections: *Data*, which is used to specify each list of solution frequencies, and *Response Frequencies*, where, once added, each list of solution frequencies will appear and can be chosen for use in an analysis.

Once a list of solution frequencies has been created, you can select it for use in any Analysis Set where the *Solution Frequencies* tab is available. Only items which are "checked" in the *Response Frequencies* list for an Analysis Set will be written to the Nastran input file.



Data

The options available to specify a list of solution frequencies correspond to the various entries Nastran has to define a frequency list (FREQ, FREQ1, FREQ2, FREQ3, FREQ4, and FREQ5). In all cases, you need to click the *Add* button to have the list appear in the *Response Frequencies* list.

Form of Frequency List

This drop-down is used to select the type of "Nastran Frequency List" to create. The inputs needed to define each type of frequency list vary based on the selected option.

The types of frequency list are:

FREQ - Can be defined by selecting a function from the *Function ID* drop-down (only X values will be used) or by clicking the *Frequency Value* button, which displays the *Frequency Response Input* dialog box. In either case, the entered values are used as the solution frequencies by Nastran.

When *Frequency Division* is set to "Custom", enter values directly into the *Frequency Value (Fi)* field. Values MUST be greater than or equal to 0.0. Click the *Add* button to add the current value in *Frequency Value (Fi)* to the list of values. Click *Update* button to change a highlighted value to the value currently in the *Frequency Value (Fi)* field. Click *Delete* button to remove the value from the list. The *Reset* button can be used to clear all values from the list. The *Copy to Clipboard* and *Paste from Clipboard* icon buttons can be used to copy/paste the current list of values to the clipboard or paste values from the clipboard into the dialog box, respectively.

When *Frequency Division* is set to "Bias", enter a *Number*, choose a type of *Bias* ("Bias Equal", "Bias at Start", "Bias at End", "Bias at Center", or "Bias at Both Ends"), enter a *Bias Factor* (if needed), and a *Range* (lowest value and highest value). Once all parameters have been specified, click the *Add* button to add values.

FREQ1 - Enter a First Frequency (F1), Number of Increments (NDF), and Frequency Increment (DF).

For example *F1* = 20.0, *NDF* = 5, and *DF* = 20.0 would produce a list with values of 20, 40, 60, 80, 100, and 120.

FREQ2 - Enter a First Frequency (F1), Last Frequency (F2), and Number of Log Interval (NF).

For example *F1* = 20, *F2* = 100, and *NF* = 4 would produce a list with values of 20, 29.907, 44.7214, 66.874, and 100.

FREQ3 - Enter First Frequency (F1), Last Frequency (F2), Number of Frequencies (NEF) (number of excitation frequencies between two modal frequencies, value includes both modal frequencies), and Cluster (factor used for "clustering" of excitation frequencies near the end points of range). Also, you can choose to use *Logarithmic* (checked) or *Linear* (unchecked) interpolation between frequencies.

For example, *F1* = 20, *F2* = 300, *NEF* = 4, and *Cluster* = 1.0, *Logarithmic* not checked, and 2 Modes with frequency values 89.8135 and 243.5258 would solve using a list of **20**, 43.271, 66.5421, **89.8131**, 141.051, 192.288, **243.526**, 262.351, 281.175, and **300**. **Bold** values are first, last, and 2 modal values.

FREQ4 - Enter First Frequency (F1), Last Frequency (F2), *Number of Frequency (NFM)* (number of evenly spaced frequencies per "spread" mode), and *Spread (FSPD, +/-%)* (specified as a % and is the "frequency spread", +/- the fractional amount, for each mode).

For example, *F1* = 20, *F2* = 300, *NFM* = 5, and *FSPD, +/-%* = 3, with 2 Modes of 89.8135 (Mode 1) and 243.5258 (Mode 2) would produce a list of 87.1188 (97% of Mode 1 value), 88.4659 (98.5%), 89.8135 (100%), 91.1603 (101.5%), 92.5075 (103%), 236.22 (97% of Mode 2 value), 239.873 (98.5%), 243.5258 (100%), 247.179 (101.5%), and 250.832 (103%).

FREQ5 - Enter First Frequency (F1) and Last Frequency (F2). Only modes which fall within the frequency range between *F1* and *F2* will be used. Now enter "fractions" of each mode to use by selecting a function from the Function ID drop-down (only X values will be used) or by clicking the Frequency Fraction button, which displays the Frequency Response Input dialog box. This dialog box is similar to the one used when creating a list for FREQ, only you enter values as Frequency Fraction (FRi) values instead of actual frequency values. Entering a value of 1.0 will create a value equal to 100% of each modal value in the specified range, 0.9 will create a value 90% of each modal value, while 1.05 will create a value 105% of each modal value.

For example, *Frequency 1* = 20, *Frequency 2* = 300, with 2 Modes of 89.8135 (Mode 1) and 243.5258 (Mode 2). Using values of 0.9, 1.0, and 1.05 for the *Frequency Fraction (FRi)* would produce a list with 80.8318 (90% of Mode 1 value), 89.8135 (100%), 94.3038 (105%), 219.173 (90% of Mode 2 value), 243.5258 (100%), and 255.702 (105%).

Add Button

Click this button to add a new list of solution frequencies to the *Response Frequencies* list.

Update Button

When an item is highlighted in the *Response Frequencies* list, it will appear in the *Data* section. You can now make changes, then click this button to update a current list of solution frequencies.

Response Frequencies

This list shows all available lists of solution frequencies currently in the model. To select a list of solution frequencies, click the check box to the left of the ID. Only "checked" items in a particular Analysis Set will be written to the Nastran input file and the "checked" items can vary from Analysis Set to Analysis Set. To "check" all available lists of solution frequencies, click the *Select All* icon button or click the *Select None* icon button to "uncheck" all.

Only one item at a time can be highlighted in the *Response Frequencies* list. When highlighted, the data for that list of solution frequencies will appear in the *Data* section and can be updated. Click the *Delete* button to delete the highlighted item or click *Delete All* to delete all lists of solution frequencies.

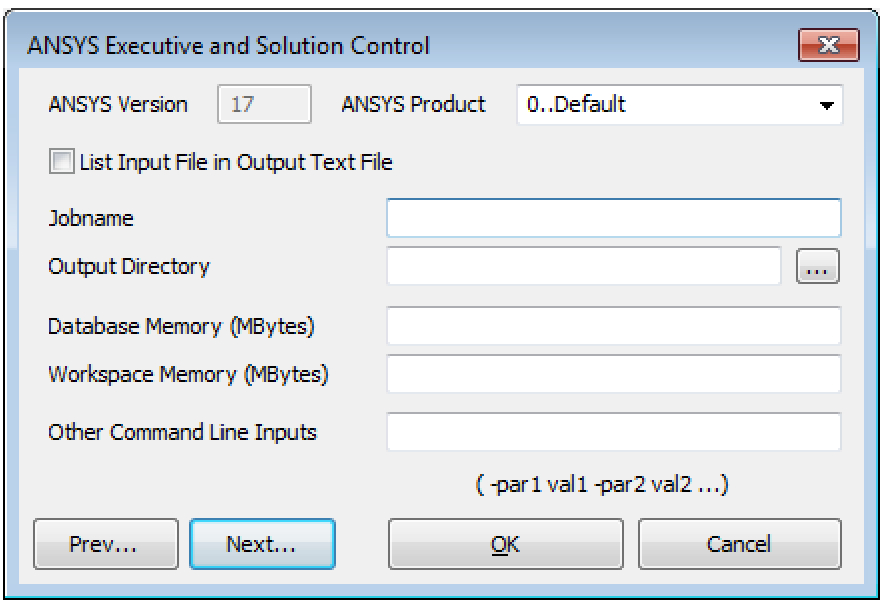
A number of bugs were corrected

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

ANSYS Interface

- Added *ANSYS Executive and Solution Control* dialog box to the Analysis Set Manager.

The ANSYS Executive and Solution Control dialog box contains information about current ANSYS Version setup to run with FEMAP and allows you to specify a number of command line arguments.



ANSYS Version

The version number displayed relates to the solver executable specified for *Ansys* on the *Solvers* tab of the *File, Preferences* command or by the ANSYS_EXE environment variable. See Section 2.6.2.6, "Solvers" or Section 4.10.2.1, "Run Analysis Using Linked Solver / VisQ / Local Settings" in the FEMAP Commands Manual for more information. If no version is defined, defaults to version 10.

ANSYS Product

Allows you to choose a particular *ANSYS Product* to launch, which may be useful to select to only use the license(s) needed for your analysis. Choose from *0..Default*, *1..ANSYS Multiphysics*, *2..ANSYS Mechanical*, *3..ANSYS Structural*, *4..ANSYS Mechanical Emag*, or *5..ANSYS Mechanical CFL-Flo*. Includes "-p" on command line.

List Input File in Output Text File

When off (default), includes "-b nolist" on command line and does not include the text of the input file in the printed output file. When on, includes "-b list" on command line to list text of input file to printed output file.

Jobname

When specified, is the initial "jobname", a name assigned to all files generated by the program for a specific model. Includes "-j (specified jobname)" on command line. Has a maximum length of 32 characters and spaces should be avoided. If not specified, assumes file name is the "jobname".

Output Directory

When specified, allows you set a location for output to be placed. The "..." icon button can be used to browse to a directory location. Includes "-dir (specified output directory)" on command line. If not specified, uses default output directory for FEMAP.

Database Memory (MBytes)

When specified as a positive value, defines the initial memory allocation for the database. If specified as a negative value, defines a fixed memory allocation for the database throughout the run. Includes "-db (value)" or "-db (-value)" on command line. If no value is specified, "-db" is omitted and default of 1024 MB is used.

Workspace Memory (MBytes)

When specified as a positive value, defines the initial overall memory allocation. If specified as a negative value, defines a fixed overall memory allocation throughout the run. Includes "-m (value) or "-m (-value)" on command line. If no value is specified, "-m" is omitted and default of 2048 MB is used.

Other Command Line Options

Allows you to specify other command line arguments. Consult ANSYS documentation for more information.

- Added support to write non-zero constraints as D, (node ID), (UX, UY, UZ, ROTX, ROTY, or ROTZ), (non-zero value). Also, added support to read non-zero values on constraints from an ANSYS input file.
- Added support to write *DOF Spring to Ground* elements as COMBIN14 elements. An additional node constrained in all six degrees of freedom will also be written to the ANSYS input file. When COMBIN14 elements are read from an ANSYS input file, *DOF Spring* elements/properties are created, not *DOF Spring to Ground* elements/properties.
- Added support to write *Spring/Damper* elements which reference a *Spring/Damper* property with *Type* set to *CBUSH* and *Spring/Damper to Ground* elements as MATRIX27 elements.

If only Stiffness is defined on the property, then it will write ET, (element ID), MATRIX27, 0, 0, 4. The property information is then written as R, (property ID) followed by RMODIF, (property ID), 1, (stiffness value) to RMODIF, (property ID), 78, (stiffness value)

If only Damping and Structural Damping values are defined on the property, then it will write ET, (element ID), MATRIX27, 0, 0, 5. The property information is then written as R, (property ID) followed by RMODIF, (property ID), 1, (damping value) to RMODIF, (property ID), 78, (damping value)

If any combination of stiffness and damping values are defined, then two MATRIX27 elements will be created, one for stiffness, on for damping. The property information is then written to R, (property ID) followed by RMODIF, (property ID), 1, (stiffness value) to RMODIF, (property ID), 78, (stiffness value) and R, (lowest unused property ID) followed by RMODIF, (lowest unused property ID), 1, (damping value) to RMODIF, (lowest unused property ID), 78, (damping value).

For *Spring/Damper to Ground* elements, an additional node constrained in all six degrees of freedom will be written to the ANSYS input file.

- Updated support when reading MATRIX27 elements and associated property information from ANSYS input file, which now become *General Matrix* elements referencing *General Matrix* properties set to the appropriate *Matrix Type* in FEMAP.

A number of bugs were corrected.

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

ABAQUS Interface

- Added support to write *DOF Spring* elements which have both Stiffness and Damping defined as *ELEMENT, TYPE=SPRING2/*ELEMENT, TYPE=DASHPOT2 combinations. When reading these entries from an ABAQUS input file, when both Stiffness and Damping are defined, two *DOF Spring* properties will be created (one containing the stiffness values, the other containing the damping values) and two *DOF Spring* elements will be created (one referencing the property with the stiffness value, the other referencing the property with the damping value).

- Added support to write *DOF Spring to Ground* elements with only Stiffness values defined as *ELEMENT, TYPE=SPRING1, with only Damping values defined as *ELEMENT, TYPE=DASHPOT1, and with both Stiffness and Damping values defined as *ELEMENT, TYPE=SPRING1/*ELEMENT, TYPE=DASHPOT1 combinations. When reading these entries from an ABAQUS input file, when both Stiffness and Damping are defined, two *DOF Spring to Ground* properties will be created (one containing the stiffness values, the other containing the damping values) and two *DOF Spring to Ground* elements will be created (one referencing the property with the stiffness value, the other referencing the property with the damping value).
- Added support to write *Spring/Damper* elements which reference a *Spring/Damper* property with *Type* set to *CBUSH* and *Spring/Damper to Ground* elements as *MATRIX INPUT/*MATRIX ASSEMBLE combinations. When read from the ABAQUS input file, *General Matrix* elements referencing *General Matrix* properties set to the appropriate *Matrix Type* will be created in FEMAP.

A number of bugs were corrected.

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

DYNA Interface

- Added *Advanced...* button to *LS-DYNA Analysis Control* dialog box, which allows the user to enter values which will write the *CONTROL_IMPLICIT_AUTO and *CONTROL_IMPLICIT_GENERAL entries when using the implicit solver.
- Added support to write non-zero constraints to the LS-Dyna input file as *BOUNDARY_PRESCRIBED_MOTION_NODE entries.
- Added “98..LS-DYNA Simplified Johnson Cook” material type to *Other Types*, which writes *MAT_SIMPLIFIED_JOHNSON_COOK.
- Added support to use a combination of beam elements with sections defined in Femap (rectangular or circular bar) with Formulation set to “9..Spotweld” and Material Type “100..LS-DYNA Spotweld” in *Other Types*. The section properties will be used to determine the required values on *BEAM_SECTION cards. If the cross section is not a rectangular or circular bar and Area>0.0, then the square root of the Area will be used for all required thicknesses. However, thicknesses may also be specified when creating or editing the “100..LS-DYNA Spotweld” material, but will only be used if the beam section property has an Area = 0.0.
- Updated the translator to not always write the *CONTROL_SOLUTION entry.
- Updated default value specified for *Termination Time* in the *LS-DYNA Analysis Control* dialog box to be 1.0.
- Updated default value specified for *Output Time Interval* in the *LS-DYNA Analysis Control* dialog box to be 0.01.
- Updated “2..LS-DYNA Orthotropic Elastic” material type in *Other Types* by adding *Shear Mod Freq Damp G* and *Limit Stress SIGF* fields.
- Updated “34..LS-DYNA Fabric” material type in *Other Types* by adding 16 new fields.
- Updated “36..LS-DYNA 3-Parameter Barlat” material type in *Other Types* by adding *Hardening Rule Load Func* field.
- Updated “54..LS-DYNA Enhanced Composite Damage” material type in *Other Types* by adding *2WAY (1=On)*, *Mat Angle MANGLE*, *Pct Failed Layers PFL*, *Dmg Init T Shear EPSF*, *Final Rupture Shear EPSR*, *Tr Shear Max Damg TSMD*, *Ortho softening SOFT2*, *Max->Min Fib T SLIMT1*, *Max->Min Fib C SLIMC1*, *Max->Min Mtx T SLIMT2*, *Max->Min Mtx C SLIMC2*, *Max->Min Shear SLIMS*, *Stress Red Cycles NCYRED*, *Trans Shear Soft SOFTG*, *Load Curve XC LCXC*, *Load Curve XT LCXT*, *Load Curve YC LCYC*, *Load Curve YT LCYT*, *Load Curve SC LCSC*, and *strnRate Avg Opt DT* fields.
- Updated “67..LS-DYNA Nonlinear Elastic Discrete Beam” material type in *Other Types* by adding *CST (0,1 or 2)* and SCOOR (-3 to 3) fields.

- Updated "68..LS-DYNA Nonlinear Plastic Discrete Beam" material type in *Other Types* by adding *CST (0,1 or 2)* and SCOOR (-3 to 3) fields.
- Updated "103..LS-DYNA Anisotropic Viscoplastic" material type in *Other Types* by adding *Fail* and *NUMINT* fields.
- Updated "126..LS-DYNA Modified Honeycomb" material type in *Other Types* by adding *VREF*, *TREF*, and *SHDFLG(0,1)* fields.

A number of bugs were corrected

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

PATRAN Interface

- Added support for reading RBE2 and RBE3 elements (MPCs) from the PATRAN Neutral File Packet 14.
- Updated support when reading PATRAN materials to support Material Types 4 (thermal isotropic) and 5 (thermal anisotropic), and added reading of specific heat and emissivity material properties.
- Added writing of non-zero constraints to PATRAN Neutral File. When reading non-zero constraints, they will come displacement loads.

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

I-DEAS Interface

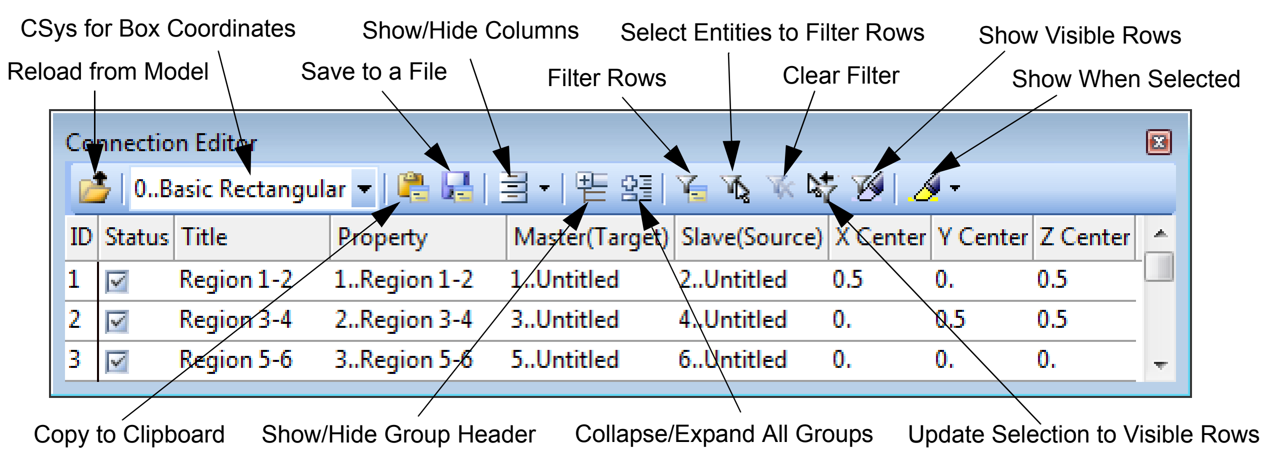
- Added support to read and write non zero constraints.

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

Tools

- Updated the *Tool, Measure, Distance Between Geometry* to allow you to measure from any entity type to an arbitrary Plane defined by the user. The *Overall Only* option is not available when *To* is set to *Plane*, therefore only the *Minimum* distance to the specified plane is reported.
- Added the *Tools, Connection Editor* command, which will display the Connection Editor dockable pane.

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



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

When the *Connection Editor* pane is floating, it can be repositioned and resized, just as any other floating dockable pane. The *Connection Editor* can be closed by clicking the "X" on the right side of the Title bar. To reopen the *Connection Editor*, use the *Tools, Connection Editor* menu.

Connection Editor Rows and Columns

All data from each Connector is sent to the *Data Table* 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 your 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 header (this will bring up a two headed arrow), then clicking and dragging the column to the desired width.

Columns can be turned on and 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 Connector. Cannot be modified directly in the table.

Status - Interactive toggle displays current status of Connector, which can be modified in the table. When checked, Connector is "Enabled" and is exported. When unchecked, Connector is "Disabled" and is not exported.

Color (column not displayed by default) - Current Color ID, in terms of the FEMAP Color Palette, of Connector. 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 Connector. 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

Title - Connector Title, which can be modified by inputting a new *Title* and pressing *Enter*/selecting another row

Property - Connection Property currently assigned to the Connector. Can be modified by selecting a Connection Property from the drop-down list or by clicking the "Select Connection Property" icon button to the right of the drop-down, then selecting a connection property from the *Select Connection Property* dialog box.

Master (Target) - Connection Region currently specified as the "Master (Target)" in the Connector. Can be modified by selecting a Connection Region from the drop-down list or by clicking the "Select Region" icon button to the right of the drop-down, then selecting a connection region from the *Select Region* dialog box.

Slave (Source) - Connection Regions currently specified as the "Slave (Source)" in the Connector. Can be modified by selecting a Connection Region from the drop-down list or by clicking the "Select Region" icon button to the right of the drop-down, then selecting a connection region from the *Select Region* dialog box.

Note: When multiple rows are highlighted and a *Color*, *Layer*, *Title*, *Property*, *Master (Target)*, or *Slave (Source)* field is edited by typing, using the drop-down, or using an icon button, you will be asked "Ok to Update All Selected Connectors (No=Update Active Only)?". If you answer *Yes*, all highlighted rows will be updated, while answering *No* will only update the row where the editing operation occurred.

X Center, Y Center, Z Center - The XYZ Coordinates of the centroid of a "box" encompassing both regions in each Connector, displayed in the Coordinate System selected in the *Connection Editor*. Cannot be modified.

X Size, Y Size, Z Size (column not displayed by default) - The X, Y, and Z distances from the "Min" coordinate of a "box" encompassing both regions in each Connector to the "Max" coordinate. Cannot be modified.

X Min, Y Min, Z Min (column not displayed by default) - The "minimum" XYZ Coordinates of a "box" encompassing both regions in each Connector, displayed in the Coordinate System selected in the *Connection Editor*. Cannot be modified.

X Max, Y Max, Z Max (column not displayed by default) - The "maximum" XYZ Coordinates of a "box" encompassing both regions in each Connector, displayed in the Coordinate System selected in the *Connection Editor*. Cannot be modified.

Data Table Icons

Reload from Model - This icon can be used to insure all Connectors in the model appear in the *Connection Editor* in their current state, should the model and the pane become out of sync, which is not typical.

CSys for Box Coordinates - This drop-down allows you to select the coordinate system used to display the coordinates in the *X Center*, *Y Center*, *Z Center*, *X Min*, *Y Min*, *Z Min*, *X Max*, *Y Max*, and *Z Max* columns.

Copy to Clipboard - Copies the current data in the *Connection Editor* to the clipboard so it can be pasted into the *Messages* window or an outside program

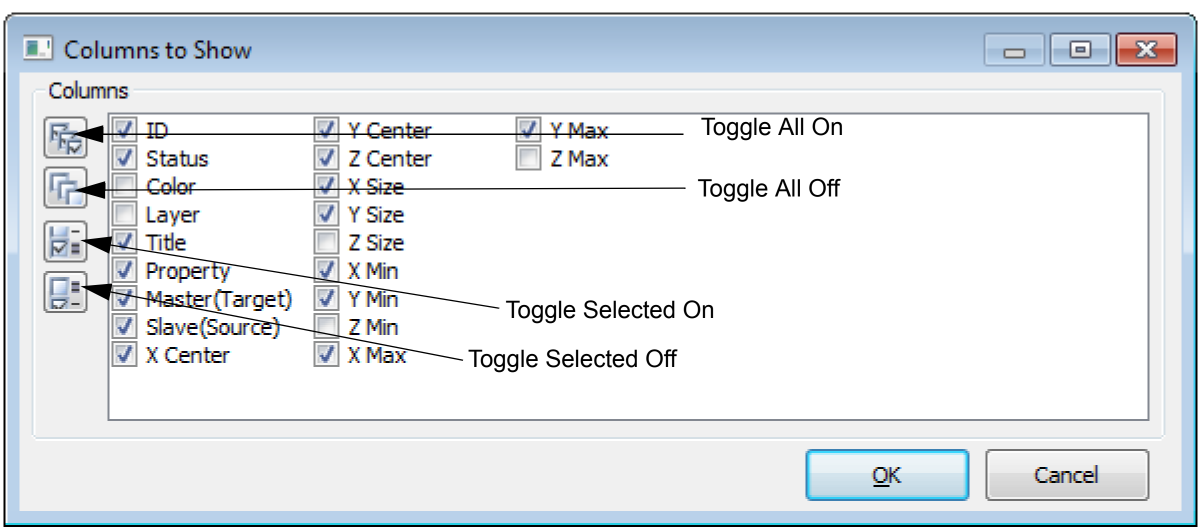
Note: Because of the Rich Text Format copied from the *Connection Editor*, you may need to do some resizing of columns in order to create a correctly formatted table when it is copied into the *Messages* pane. When copying tables from the *Connection Editor*, there should be no issues when pasting into outside programs which support Rich Text Format, such as Microsoft Word.

Note: Ctrl+C can be used as a general copy command in FEMAP. FEMAP takes into account which window or dockable pane is currently active. When the *Connection Editor* pane is active, Ctrl+C will copy all rows currently in the table to the clipboard.

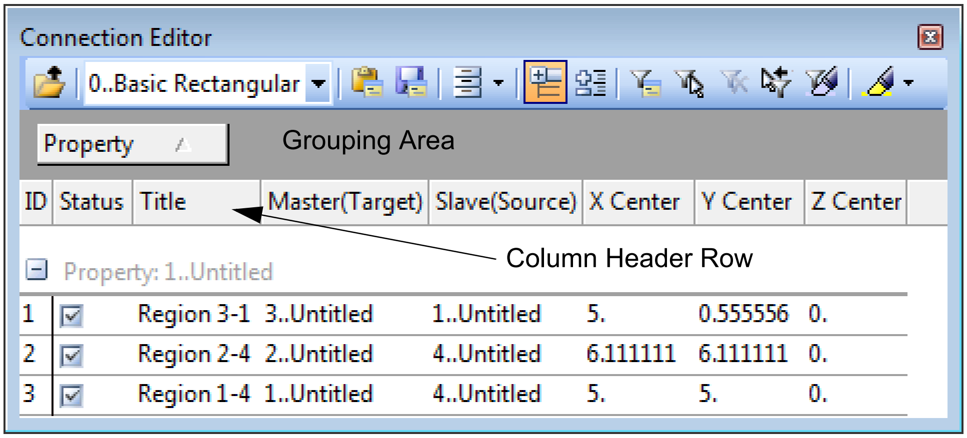
Sa**ve to a File** - Allows you to save the contents of the *Connection Editor* to a text file. You can save the data as Tab Delimited Text (*.TXT file), Comma-Separated Text (*.CSV file), or in Rich Text Format (*.RTF file).

H**ide/Show Columns Menu** - Clicking this icon brings up a menu with commands and a listing of all available columns (visible columns are marked with a check mark in an orange box). By highlighting a column and clicking the mouse, you can toggle the column 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 *Connection Editor*



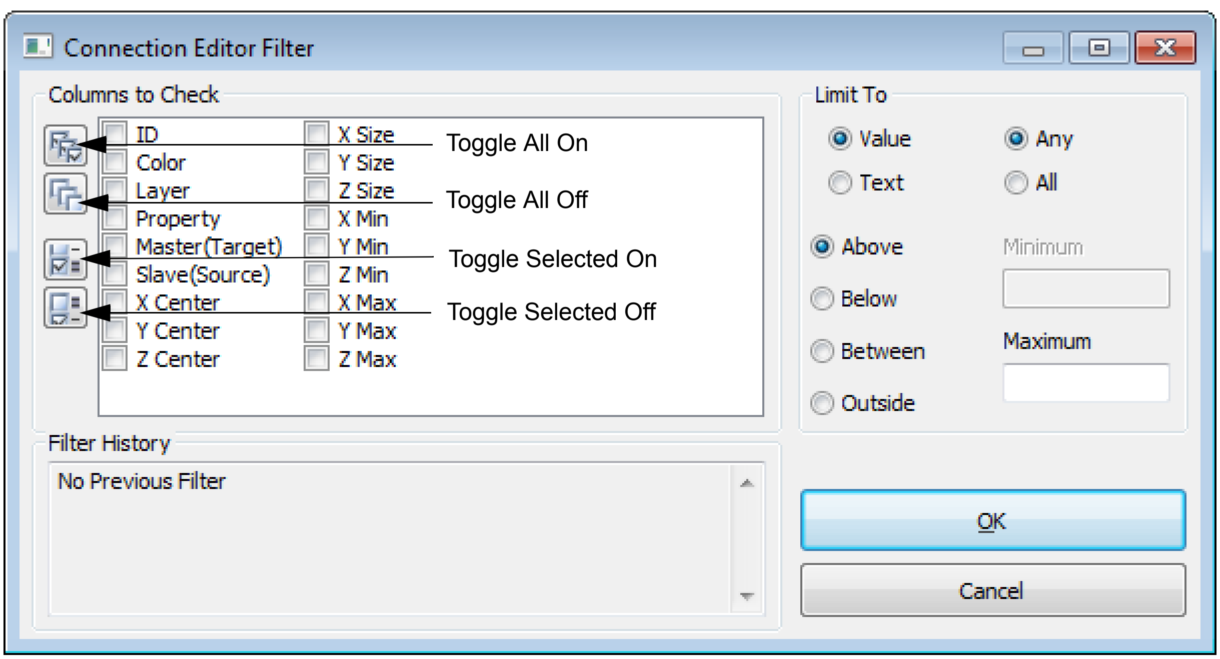
Show/Hide Group Header - When on (icon highlighted by orange background), the *Connection Editor* allows you to drag column headers up into a specified area. Once there, the *Connection Editor* will partition the rows of Connector data using a specific group header or set of group headers. This allows you to "group" Connectors in the table with similar characteristics (such as the same connection *Property*, *Master (Target)* region, *Color*, etc.) together for a specific purpose. The grouping is multi-level, therefore the Connectors 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 Connection Editor all at once. Can be used to reduce the complexity of the table and focus on specific Connectors

Filter Rows - Brings up the *Connection Filter* dialog box.Allows you to choose columns to check for specific values or text.



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 *Connection Editor.*

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

Clear Filter - Clears all filter settings and all Connectors previously shown in the *Connection 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 *Connection Editor.* This command can be used when the filter is on or off, as well as, when there are groups in the *Connection Editor.* This command will completely purge the selection list of all other entity types other than Connectors in the *Connection Editor*, so be careful about using it if you have created a large selection list with multiple entity types.

Show Visible Rows - Highlights (*Window, Show Entities* command) all Connectors that are currently visible as rows in the *Connection Editor.*

Show When Selected Menu - Contains a number of options to "show" the Connectors currently highlighted in the *Connection Editor* by "highlighting/showing" them in the main graphics window. By default, this command is set to off. The commands on this menu use different options found in the *Window, Show Entities* command. 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 Connectors 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 Connectors NOT associated with the selected entities in the *Connection Editor* to temporarily disappear until *Show When Selected* is toggled off or the model is regenerated.

The *Show Labels* and *Show Normals* (not used when highlighting Connectors) 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 Connection Features

Expand/Collapse toggles - Only appear when using "Groups" in the *Connection Editor.* Allow you to collapse and expand individual group categories and sub-categories inside the *Connection 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:

Edit... - This command only appears at the top of the context-sensitive menu when the cursor is over a field in the Property, Master (Target), or Slave (Source) columns. When Edit Connection Property (#) is available, the Modify, Edit, Connection Property command is run. When Edit Connection Region (#) is available, the Modify, Edit, Region command is run. For more information, see Section 4.8.2.1, "Modify, Edit Menu".

New Connector - Allows you to create a new Connector using the *Connect, Connector* command. For more information, see Section 4.4.5, "Connect, Connector... (Contact Pair)".

New Connection Regions - Allows you to create a new Connection Regions using the *Connect, Connection Regions* command. For more information, see Section 4.4.4, "Connect, Connection Region...".

New Connection Property - Allows you to create a new Connection Property using the *Connect, Connection Property* command. For more information, see Section 4.4.3, "Connect, Connection Property...".

Show Connector (# Selected) - "Shows" (highlights in the graphics window) the Connector(s) corresponding to the row(s) currently highlighted in the *Connection Editor.*

Show Master (Target) - "Shows" (highlights in the graphics window) the "Master (Target)" Connection Region(s) corresponding to the row(s) currently highlighted in the *Connection Editor.*

Show Slave (Source) - "Shows" (highlights in the graphics window) the "Slave (Source)" Connection Region(s) corresponding to the row(s) currently highlighted in the *Connection Editor.*

Retitle - Will update the *Title* of each Connector currently highlighted in the *Connection Editor* using the format "Region 'Master (Target) Region ID'-'Slave (Source) Region ID'".

Reverse - Reverses the *Slave (Source)* and *Master (Target)* Connection Regions of each Connector currently highlighted in the *Connection Editor*. In other words, it makes the region designated the *Master (Target)* the *Slave (Source)* and the *Slave (Source)* the *Master (Target)*.

Delete Connectors - Deletes the Connector(s) currently highlighted in the *Connection Editor*.

Delete All - Attempts to delete all "components" of each Connector (i.e., Connector, Connection Regions, and Connection Property) currently highlighted in the *Connection Editor*. Connection Regions or Connection Properties which are used by Connectors not currently highlighted, will not be deleted.

Select All - Selects all of the rows currently displayed in the *Connection Editor*. Filtered rows are not selected.

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*.

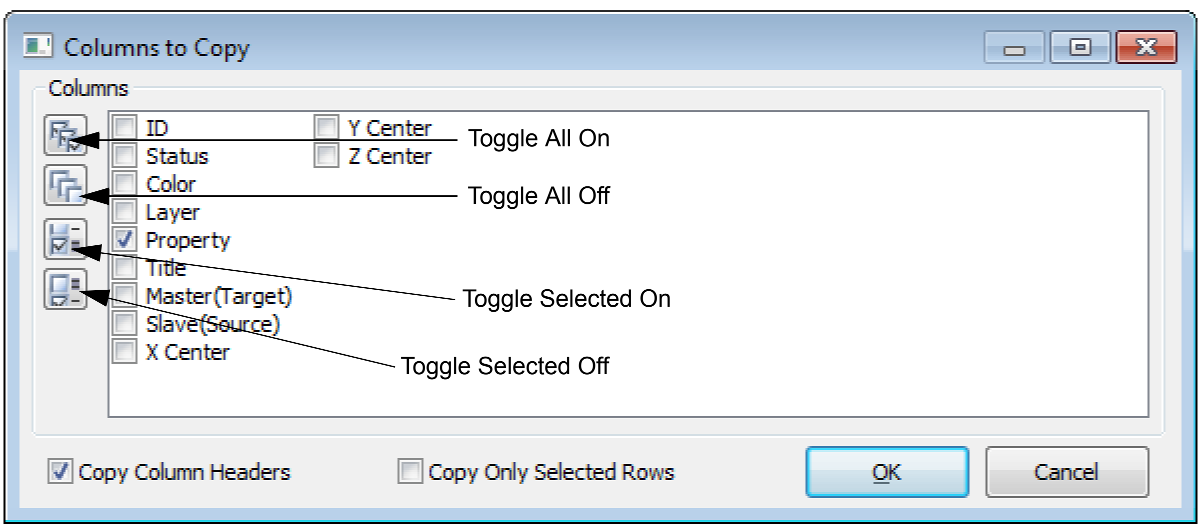
Note: It is possible to choose multiple entity rows at the same time and perform operations on all of the rows at once. In order to choose multiple entity rows, hold down the CTRL key and select individual entities with the mouse or hold down the SHIFT key and select a first row and a last row and all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

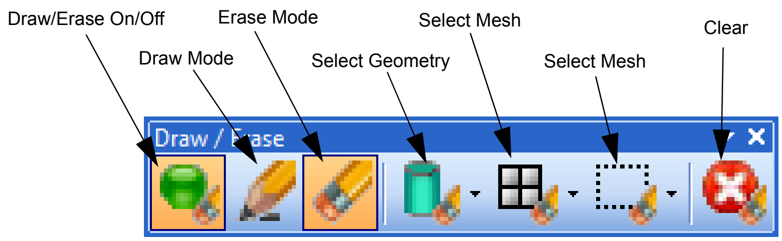
Context Sensitive Menu (Other) - When the cursor is not positioned over a column header or a row, right mouse clicking will bring up a context sensitive menu. This menu includes the *New Connector*, *New Connection Region*, *New Connection Property*, and *Delete All* commands from the context-sensitive menu for "Rows" and also offers:

Automatic... - Runs the *Connect, Automatic* command (see Section 4.4.1, “Connect, Automatic...”)

Select... - Runs the Connect, Surfaces command (see Section 4.4.2, “Connect, Surfaces...”)

Delete All Connectors - Deletes all Connectors in model.

- Added the *Tools, Toolbars, Draw/Erase* command to display the *Draw/Erase* Toolbar.



This toolbar offers two different modes, one which will temporarily only “Draw” entities of certain types selected with the toolbar, while the other will temporarily “Erase” the selected entities. The selected entities will remain “Drawn” or “Erased” until cleared using the *Clear* icon on the toolbar. At any time you can switch from “Draw” mode to “Erase” mode and “swap” what appears in the graphics window. The overall functionality of the *Draw/ Erase* toolbar can also be toggled off at any time to display all entities currently visible in the model, then toggled on again.

Entities are selected using any combination of commands on the *Select Geometry*, *Select Mesh*, and/or *Select Area* icon menus. When the list of selected entities is cleared, which can be done at any time, all entities currently visible in the model will be displayed, then different entities can be selected.

If an entity is not visible before using the functionality of *Draw/Erase* Toolbar, it will not appear in either “Draw” mode or “Erase” mode. An entity may not be visible before using *Draw/Erase* for any of the following reasons:

First, overall visibility of an entity type is turned “off”. This can be fully controlled using the *Entity/Label* tab of the *View, Visibility* command or partially controlled using the *Entity Display* toolbar.

Second, individual entity visibility for certain entity types is turned “off”. This can be controlled using the various “entity” tabs in the *View, Visibility* command, the “visibility check boxes” in the *Model Info* tree, or the *Visibility* portion of various context-sensitive menus when using the *Select* toolbar.

Third, Groups and/or Layers in use and an entity is not any visible Group and/or on any visible Layer. This can be controlled by the *Group* tab and the *Layer* tab of the *View, Visibility* command or the *Group* and *Layer* branches in the *Model Info* tree.

Note: The “Erase” mode uses an “list of erased entities”. When you switch to “Draw” mode, this list is simply inversed, so all available entities which were not in the “erase list” are in the “draw list”.

Draw/Erase on/off

This icon can be used to toggle on/off the overall functionality of the *Draw/Erase* toolbar. When “on”, which is the default, “Draw/Erase” functionality is enabled and the icon will appear “highlighted” and display a “Green Light”. When “off”, “Draw/Erase” functionality is disabled, therefore all entities currently visible in the model will be displayed and the icon will no longer be highlighted and display a “Red Light”.

Draw Mode

This icon is used to select “Draw” mode. The “Draw” mode is designed to display only the entities selected using the commands on the *Select Geometry*, *Select Mesh*, and/or *Select Area* icon menus, from the entities currently visible in the model. The default behavior of “Draw” mode can be altered by turning off the *Auto Select Mesh* option on the *Select Geometry* icon menu, selecting an option other than *Erase All Selected* on the *Select Area* icon menu, and/or turning off the *Exclusive Draw Mode* option on the *Select Area* icon menu. These options are discussed in greatly detail below.

Note: When selecting entities in “Draw” mode, *Draw/Erase on/off* will be set to “off” automatically to enable you to select from all entities currently visible in the model.

Erase Mode

This icon is used to select “Erase” mode. The “Draw” mode is designed to “erase” (turn visibility “off”) the entities selected using the commands on the *Select Geometry*, *Select Mesh*, and/or *Select Area* icon menus, from the entities currently visible in the model. The default behavior of “Erase” mode can be altered by turning off the *Auto Select Mesh* option on the *Select Geometry* icon menu, selecting an option other than *Erase All Selected* on the *Select Area* icon menu, and/or turning off the *Exclusive Draw Mode* option on the *Select Area* icon menu. These options are discussed in greatly detail below.

Select Geometry menu

This icon menu is used to choose the type of geometric entity to temporarily “Draw” or “Erase”. Any additional entities which will be “Drawn” or “Erased” depends on the type of geometric entity being selected and the current setting of the *Auto Select Mesh* option (“on” is the default).

Select Solids... - When selecting *Solids*, all *Surfaces*, *Curves*, and *Points* referenced by the Solid(s) will also automatically be “Drawn” or “Erased”. In addition, if ALL geometric entities referenced by a geometry-based *Region*, *Load*, and/or *Constraint* are also referenced by the selected *Solid(s)*, they will also be “drawn” or “erased”.

When the *Auto Select Mesh* option is enabled, any elements in the *Solid(s)* or on any *Surface*, *Curve*, or *Point* referenced by the *Solid(s)* will also be “drawn” or “erased”, along with the nodes on those elements. In addition, if ALL mesh entities in a *Region* are also mesh entities associated with the *Solid(s)* or any *Loads* and/or *Constraints* are located on mesh entities associated with the *Solid(s)*, they will also be “drawn” or “erased”.

Select Surfaces... - When selecting *Surfaces*, all *Curves* and *Points* referenced by the *Surface(s)* will also automatically be “Drawn” or “Erased”. In addition, if ALL geometric entities referenced by a geometry-based *Region*, *Load*, and/or *Constraint* are also referenced by the selected *Surface(s)*, they will also be “drawn” or “erased”.

When the *Auto Select Mesh* option is enabled, any elements on the *Surface(s)* or on any *Curve* or *Point* referenced by the *Surface(s)* will also be “drawn” or “erased”, along with the nodes on those elements. In addition, if ALL mesh entities in a *Region* are also mesh entities associated with the *Surface(s)* or any *Loads* and/or *Constraints* are located on mesh entities associated with the *Surface(s)*, they will also be “drawn” or “erased”.

Select Curves... - When selecting *Curves*, all *Points* referenced by the *Curve(s)* will also automatically be “Drawn” or “Erased”. In addition, if ALL geometric entities referenced by a geometry-based *Region*, *Load*, and/or *Constraint* are also referenced by the selected *Curve(s)*, they will also be “drawn” or “erased”.

When the *Auto Select Mesh* option is enabled, any elements on the *Curve(s)* or on any *Point* referenced by the *Curve(s)* will also be “drawn” or “erased”, along with the nodes on those elements. In addition, if ALL mesh entities in a *Region* are also mesh entities associated with the *Curve(s)* or any *Load* and/or *Constraint* are located on mesh entities associated with the *Curve(s)*, they will also be “drawn” or “erased”.

Select Points... - When selecting *Points*, only the selected Point(s) will be “Drawn” or “Erased”. In addition, if ALL geometric entities referenced by a geometry-based *Loads* and/or *Constraints* are also referenced by the selected *Point(s)*, they will also be “drawn” or “erased”.

When the *Auto Select Mesh* option is enabled, any elements on the *Point(s)* will also be “drawn” or “erased”, along with the nodes on those elements. In addition, if any *Loads* and/or *Constraints* are located on mesh entities associated with the *Point(s)*, they will also be “drawn” or “erased”.

Auto Select Mesh - When on, mesh associated with the selected geometric entity will also be “drawn” or “erased”. See the various “Select...” sections above for more detail.

Note: If an element is not associated with a geometric entity then it will not automatically be considered for “Draw” or “Erase”, even if all of the element’s nodes are shared with an element associated with a geometric entity being “drawn” or “erased”. The element would have to be selected separately.

Select Mesh menu

This icon menu is used to choose various type of entities to temporarily “Draw” or “Erase”. Any additional entities which will be “Drawn” or “Erased” depends on the type of entity being selected.

Note: If the *Nodes On* option found on the *Element*, *Material*, and *Property* tabs of the *Visibility* dialog box is enabled, then nodes of non-visible elements will remain visible. See Section 6.1.4, “View, Visibility...”

Select Elements... - When selecting *Elements*, any node associated with a visible element will also be visible. If a node is only associated with elements which are not currently visible, due to *Draw/Erase*, then that node will also not be visible. In addition, if ALL nodes or elements in a *Region* are also associated with the *Element(s)* or any *Loads* and/or *Constraints* are located on mesh entities associated with the *Element(s)*, they will also be “drawn” or “erased”.

Select Properties... - When selecting *Properties*, all elements which reference each *Property* will also automatically be “Drawn” or “Erased”, along with the nodes on those elements. In addition, if ALL nodes or elements in a *Region* are also associated with the *Properties* or any *Loads* and/or *Constraints* are located on mesh entities associated with each *Properties*, they will also be “drawn” or “erased”.

Select Materials... - When selecting *Materials*, all elements which reference each *Material* will also automatically be “Drawn” or “Erased”, along with the nodes on those elements. In addition, if ALL nodes or elements in a *Region* are also associated with the *Materials* or any *Loads* and/or *Constraints* are located on mesh entities associated with each *Materials*, they will also be “drawn” or “erased”.

Select Coord Sys... - When selecting *Coordinate Systems*, only *Coordinate Systems* will be “Drawn” or “Erased”.

Select Regions... - When selecting *Regions*, only *Regions* will be “Drawn” or “Erased”.

Select Connectors... - When selecting *Connectors*, all *Connection Regions* associated with each *Connector* will also automatically be “Drawn” or “Erased”.

Note: If both *Connection Regions* used by a *Connector* are not visible, then the *Connector* will not be visible.

Select Area menu

This icon menu contains a variety of commands which allow you to:

Select entities using “area picking” (*Box*, *Circle*, *Polygon*, and *Freehand*)

Use functionality of the *Selector* (*Select* toolbar) to select eleven different entity types (*Selector Erase* menu)

Add a layer of entities to or remove a layer of entities from the currently selected entities (*Grow* and *Shrink*)

Choose if geometry, mesh, or both should be considered (*Erase All Selected*, *Erase Geometry Only*, or *Erase Mesh Only*)

Limit what is automatically selected when in “Draw” mode (*Exclusive Draw Mode*)

Make the “erase list” the “draw list” and vice versa (*Reverse*)

Create a new group or modify an existing group using the currently selected entities (*Create Group*)

Add entities to the list of currently selected entities by selecting a group (*Load Group*)

Box - Creates a rectangular selection box to pick multiple entities of various types to “Draw” or “Erase”.

Circle - Creates a circular selection box to pick multiple entities of various types to “Draw” or “Erase”.

Polygon - Allows you to create a selection area by creating a multi-sided shape using a click of the mouse to end one “side” of the polygon and begin the next “side”. The polygon can have as many sides as you would like and selects multiple entities of various types to “Draw” or “Erase”.

Freehand - Allows you to create a selection area by holding down the left mouse button and “drawing” a freehand sketch. Works very similar to the *Polygon* mode, but the shape is not limited to the use of only straight lines.

Note: All for of the area selection methods can be used in conjunction with the *Pick Any Inside* and *Pick All Inside* commands found on the “quick access menu”. *Pick Any Inside* (option “off”) or *Pick All Inside* (option “on”) can also be set using he *Pick All Inside* option in the *User Interface* tab of the *File, Preferences* command. For more information, see Section 2.6.2.4, “User Interface”.

Selector Erase menu - Allows you to use the functionality of the *Selector* (i.e., the *Select* toolbar) to select *Points*, *Curves*, *Surfaces*, *Solids*, *Regions*, *Connectors*, *Coordinate Systems*, *Elements*, elements referencing *Materials*, elements referencing *Properties*, or elements referencing *Layups* to “Draw” or “Erase”. The selected entity type becomes the active entity type in the selector, allowing you to select entities without using the standard entity selection dialog box.

Grow - Adds the entities surrounding the currently selected entities to the selection list. In general, it will start with the selected entities and expand out one layer of entities in all directions to non-selected entities. The selection will move out an additional layer of entities every time the *Grow* command is used.

Shrink - Removes the outer-most layer of selected elements from the selection list. The selection will move in an additional element layer every time the *Shrink* command is used. Basically, it is the works in the reverse direction as the *Grow* command.

Erase All Selected - When selected, “Draw” and “Erase” considers both geometric entities and mesh entities. This is the default mode.

Erase Geometry Only - When selected, “Draw” and “Erase” considers only geometric entities.

Erase Mesh Only - When selected, “Draw” and “Erase” considers only mesh entities.

Exclusive Draw Mode - When on, which is the default, only the entities you have been selected to “Draw” will be visible. When off, only entities of the same entity type will be removed from the display, while entities of other entity types which are visible, will remain visible.

Reverse - Simply coverts the current “erase list” to the “draw list” and vice versa, but only for entity types which were originally selected to “erase” or “draw”. For instance, if elements were “erased”, but no geometric entities were also “erased”, then only the list of elements will be reversed and geometry visibility will remain unchanged.

Create Group - Displays the *Save Selection as Group* dialog box. Allows you to *Create New Group* containing only the entities which are currently visible. You can also use *Add to Group* (adds visible entities), *Remove from Group* (removes visible entities), or *Exclude from Group* (excludes visible entities) to modify the group selected with the drop-down (default is active group).

Load Group - Depending on the selected mode, allows you to add the entities of any existing group to the “draw list” or the “erase list”

Clear

This icon is used remove all entities currently in the “erase list” and “draw list”. Once cleared, the *Draw/Eras*e toolbar no longer effects visibility of any entity, therefore, all entities which are currently visible in the model based on overall entity type visibility, individual entity visibility, inclusion in visible Group(s), and/or inclusion on visible Layer(s) will be displayed.

OLE/COM API

- Added checking at startup and warning messages to indicate whether the API/COM server is properly registered for FEMAP.

New and modified API Objects and Attributes

- Added Draw/Erase (feDrawErase) object to the API. Also, added Enable, EraseMode, AutoSelectMesh, ExclusiveDrawMode, EraseGeometry, and EraseMesh attributes to the Draw/Erase Object.
- Added Frequency (feFreq) object to the API. Also, added type, F1, F2, DF, FSPD, NF, and LOG attributes to the Frequency Object.
- Added View Orient (feViewOrient) object to the API. Also, added Center, vCenter, Magnification, ModelAlwaysInFront, AutoAspectRatio, AspectRatio, Perspective, PerspectiveAngle, RotationAxisOption, RotationCenterOption, RotationCoordSys, SingleAxis, RotationCenter, vRotationCenter, RotationVector, and vRotationVector attributes to the View Orient Object.
- Added NasDynFreqID, NasExecGPUOpt, LinkedSolverOption, NasNXStratPlasalg, NasExecUserCmdLine, NasRandCorrOn, NasRandCorrLagInterval, NasRandCorrLagStart, NasRandCorrLagMax, vNasRandXYRequest2, NasRandXYAuto, NasRandXYPsdf, NasNonlinAdapt, AnsCmlOn, AnsCmlVersion, AnsCmlPid, AnsCmlProd, AnsCmlList, AnsCmlJobname, AnsCmlDir, AnsCmlDBmem, AnsCmlWSmem, and AnsCmlUser attributes to the Analysis Manager Object
- Added NonZeroConstraint, value, and vvalue attributes to the BCNode Object.
- Added ScaleInherit and MasterScale attributes to the Chart Object.
- Added StudyID attribute to the Chart Data Series Object.

- Added SpringUseLocation, SpringLocation, SpringUsePropLocation, SpringUseCID, SpringNoOrient, SpringCID, and SpringUsePropCID attributes to the Element Object.
- Added NastranQuadAROn, NastranQuadARLimit, NastranTriaAROn, NastranTriaARLimit, NastranTriaEPLROn, and NastranTriaEPLRLimit attributes to the Element Quality Object.
- Updated SumComponents, vSumComponents, Group, DisplayMode, ShowTotalVec, ShowNodalVec, SumContributions, and vSumContributions attributes to the Freebody Object.
- Added RadViewFactorZTOL and bRadViewFactorZTOL attributes to the Load Set Object.
- Added NumberOfLoads and NumberOfConstraints attributes to the Node Object.
- Added nas_case and nas_revision attributes to the OutputSet Object.
- Added attrVertexLoopsAsHardPoints and attrMaxQuads to the Surface Object.
- Added PerspectiveAngle, vContourVecOn, ContourVecAutoVec, ContourVecAutoOrient, TransformDeformXInput, TransformDeformYInput, TransformDeformZInput, TransformPlateTolerance, ContourVecMinVecOn, ContourVecMinVec, ContourVecMinLen, ContourVecColor1, and LabelGroup to the View Object.

Removed API Objects and Attributes

- Removed LaunchWithVisQ,vNasDynNoFreq, vNasDynLogInterp, vNasDynFreqType, vNasDynMinFreq, vNasDynMaxFreq, vNasDynSpreadCluster, NasDynNoFreq, NasDynLogInterp, NasDynFreqType, NasDynMinFreq, NasDynMaxFreq, and NasDynSpreadCluster attributes from the Analysis Manager Object.
- Removed PerspectiveDist from the View Object.

New API Methods

- Added AllFreqOn, AllFreqOff, TurnOnFreq, and TurnOffFreq to the Analysis Manager object.
- Added GetAll and PutAll to the BCEqn object.
- Added SetFixed, SetPinned, SetNoRotation, and SetArbitrary to the BCGeom object.
- Added AddNonZero to the BCNode object.
- Added RemoveAllDataSeries to the Chart object.
- Added BoundingBox and BoundingBoxInCSys to the Connection object.
- Added BoundingBoxInCSys to the Connection Region object.
- Added IsMergeable and BoundingBoxInCSys to the Curve object.
- Added ClearAll, Clear, SaveGroup, LoadGroup, Grow, Shrink, HasErased, HasErasedType, EraseSet, and GetErased to the Draw/Erase object.
- Added GetAllArray3, GetSpringOrient, SetSpringOrient, GetSpringOffset, and SetSpringOffset to the Element object.
- Added GetNastranQuadAR, NastranQuadAR, GetNastranTriaAR, NastranTriaAR, GetNastranTriaEPLR, and NastranTriaEPLR to the Element Quality object.
- Added AddFreq, UpdFreq, AddFreqByFuncID, UpdFreqByFuncID, AddFreq1, UpdFreq1, AddFreq2, UpdFreq2, AddFreq3, UpdFreq3, AddFreq4, UpdFreq4, AddFreq5, UpdFreq5, AddFreq5ByFuncID, UpdFreq5ByFuncID, AllOn, AllOff, TurnOn, and Turn Off to the Frequency object.
- Added AddOpt and AddOpt2 to the Group object.
- Added GetOutputDataSurface and MapOutputDataToLocation to the Map Output object.
- Added GetValueArray, PutValueArray, PutValue, CopyArray, GetLibraryOfType to the Material object. Also, updated GetLibrary.
- Added GetValueArray, PutValueArray, PutValue, CopyArray, GetLibraryOfType to the Property object.
- Added SendToDataTable, SetSubcase, and SetRevision to the Results Browsing object.

- Added AddSolidElementsAlongVector, AddVisible, AddConnectedElements, AddAllConnectedElements, AddConnectedFillets, AddTangentSurfacs, AddConstrained, AddLoaded, AddComponentOutputVectors, AddSimilarOutputVectors, AddComplexOutputVectors, RemoveRule, and RemoveSetRule to the Set object.
- Added GetSet to the Sort object.
- Added BoundingBoxInCSys, RemoveMeshPoint, and RemoveAllMeshPoints to the Surface object.
- Added SetEyeDirection, GetEyeDirection, ViewRotation, ModelRotation, CoordSysRotation, VectorRotation, and RollThru to the ViewOrient object.

The following functions have been added or updated:

- feAppUndoCheckpoint
- feSurfaceGetUnderlying
- feGroupGenElemShape
- feEntitySetVisibility
- feEntityGetVisibility
- feVectorArrayTransform
- feSurfaceFromMesh2
- feSolidRemoveCurve
- feSurfaceOffset
- feCrossSectionReport
- feGroupGenVisible
- feCurveSolidToFEMAP
- feResultsToDataTable
- feResultsRankingToDataTable
- feDeleteOutputEntry
- feDeleteOutput2

The following functions have been removed

- feFileReadIdeas

New and updated Global Variables

- Added Pref_UserToolsPath, Pref_NonmanifoldBooleans, Pref_RepeatCreate, Pref_ZoomAroundCursor, Pref_UseSubcaseIdForOutput, Pref_AutoConvertOdb, Pref_RenderFastPickVisible, Pref_FBDefContributions, vPref_FBDefContributions, Pref_FBDefDispMode, Pref_FBDefNodalVectorMode, vPref_FBDefNodalVectorMode, Pref_FBDefTotalVectorMode, vPref_FBDefTotalVectorMode, Pref_FBDefComponents, vPref_FBDefComponents, Pref_ElemQualQuadAR, Pref_ElemQualTriaAR, Pref_ElemQualTriaEPLR, Pref_ElemQualQuadARVal, Pref_ElemQualTriaARVal, Pref_ElemQualTriaEPLRVal, Pref_ResetNextID, Pref_SolNXNastProgram, Pref_SolMSCNastProgram, Pref_SolNEINastProgram, Pref_SolAnsysProgram, Pref_SolAbaqusProgram, Pref_SolLSDynaProgram, Pref_SolNXNastCmd, Pref_SolMSCNastCmd, Pref_SolNEINastCmd, Pref_SolAnsysCmd,Pref_SolAbaqusCmd, Pref_SolLSDynaCmd, Pref_SolEchoCmd, Pref_NasUesTrackRevsion, and Pref_RotateAroundCursor to set various preferences.
- Updated Pref_KeepSolverWindow, Pref_MinimizeDuringSolve, Pref_AnalysisProg, Pref_SolverMemory, Pref_ScrollBackLines, and Pref_WtmassFactor to set various preferences and Info_NextID and vInfo_NextID.

Preferences

Views

- Removed *2D Tensor Plot View Options Override* option, as it is no longer needed.

Graphics

- Updated *Performance Graphics* in the *Graphics Options* section. Rigid Elements (RBE2, RBE3, and RSPLINE) are now supported in performance graphics. Also, the FEMAP will now turn off Performance Graphics automatically when a mode or command is not supported, then return to Performance Graphics after leaving the mode or no longer in that command.
- Added *Fast Pick Visible* to *Graphics Options* section.

When selected, picking is enhanced to allow “Pick Front” to be used in conjunction with area picking (*Box*, *Circle*, *Polygon*, or *Freehand*). Also, the overall performance of picking when using “Pick Front” is improved.

Note: This method does have limitations. For instance, it will only pick entities that are “visible”, meaning they have at least one pixel on the screen, thus it is possible that very small elements will not be selected. Also, nodes and points can be erroneously selected even if their actual location is behind another entity of the same type. Finally, if two or more entities are fully coincident, only one of the entities can be selected from the screen while in *Pick Front*. You must switch to *Pick Normal* or *Pick Query* or turn off this option in order to be able to select coincident entities one at a time from the screen.

Note: A command, *Pick Visible*, to quickly toggle *Fast Pick Visible* on/off may be added to any menu or toolbar using *Tools, Toolbars, Customize*. This command is located in the *Additional Commands* category on the *Commands* tab of the *Customize* dialog box.

- Added *TDR Protection* to *Advanced/Debug Options* section.

This preference should only be used if you see an error which specifically mentions TDR. TDR is the Microsoft acronym for Timeout Detection and Recovery. The default value for TDR is 2 seconds, so if the GPU has not “talked” to the CPU side of the graphics driver in 2 seconds, the Operating System assumes the graphics driver has exited unexpectedly or become unresponsive, thus it reboots the graphics driver. If this occurs when using Performance Graphics, this can cause all graphics-related functionality in FEMAP to stop working and this functionality cannot be recovered without restarting FEMAP. A value of 0, which is the default, is special and instructs FEMAP to use the “system value” for “TDR Delay”, while entering a value other than 0 will specify a temporary value that only FEMAP will use for “TDR Delay”. When determining a *TDR Protection* value, be sure the value is higher than the number of processors on your graphics card.

Note: Through testing, a value in the range of 1200 to 120000 was determined to be reasonable for a “medium-level graphics card”, but these values may not be appropriate for all graphics cards.

User Interface

- Added *Dynamic Zoom Around Cursor Location* to *Mouse Interface* section.

When on, scrolling the mouse middle mouse wheel or clicking and dragging the mouse with the *Shift* key held down will zoom in/out around the location of the cursor. When off, which is default, zooming in/out by scrolling the mouse wheel or clicking and dragging the mouse with the *Shift* key held down will occur around the center of the graphics window.

Note: A command, *Zoom Around Cursor Location*, to quickly toggle *Dynamic Zoom Around Cursor Location* on/off may be added to any menu or toolbar using *Tools, Toolbars, Customize*. The command is located in the *Additional Commands* category on the *Commands* tab of the *Customize* dialog box.

- Added *Dynamic Rotate Around Cursor Location* to *Mouse Interface* section.

When on, rotates around an automatically selected location on the model near the current location of the cursor. When off, which is default, rotation will occur around the center of the view or rotation center.

Note: A command, *Rotate Around Cursor Location*, to quickly toggle *Dynamic Rotate Around Cursor Location* on/off may be added to any menu or toolbar using *Tools, Toolbars, Customize*. The command is located in the *Additional Commands* category on the *Commands* tab of the *Customize* dialog box.

- Changed Tooltip Duration to Duration in the *Graphical Selection* section.

Database

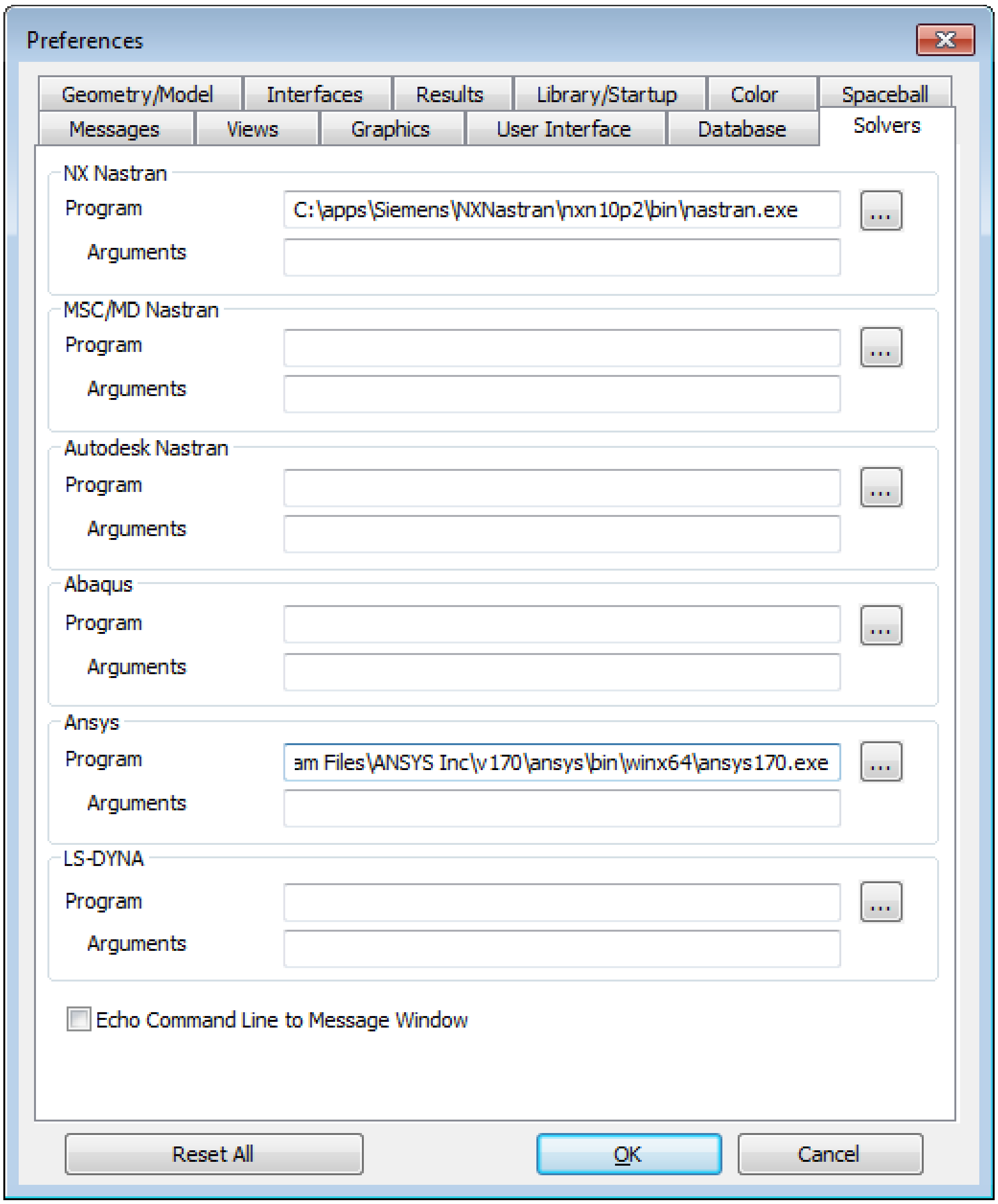
- Added *Reset Next ID after Delete All* to *Database Options* section.

When enabled, which is the default, this option will return the "Next ID" of certain entity types to "1" after the last entity of that type has been deleted from the model. The entity types tracked by this option are Point, Curve, Surface, Solid, Volume, Coordinate System, Node, Element, Material, Property, Connection Property, Connection Region, Connector, Aero Panel/Body, Aero Property, Aero Spline, Aero Control Surface, Load Set, Constraint Set, Group, Text, View, and Output Set.

Solvers - New for 11.3

- Added *Solvers* tab to specify the location of solver programs and optionally enter command line options.

The *Solvers* tab allows you to specify a location of a *Program* (i.e., finite elements solver executable) and any command line *Arguments* which should be included when the solver program is automatically launched by FEMAP



Sections are available to set up *NX Nastran*, *MSC Nastran*, *Autodesk Nastran*, *Abaqus, Ansys*, and/or *LS-DYNA*. You can also use the *Browse* button to search for a specific directory where an executable might be found.

A typical path for standard *NX Nastran* might be: C:\apps\Siemens\NXNastran\nxn10p2\bin\nastran.exe

A typical path for "ILP" *NX Nastran* might be:C:\apps\Siemens\NXNastran\nxn10p2\bin\nastran64L.exe

A typical path for *Ansys* might be: C:\Program Files\ANSYS Inc\v170\ansys\bin\winx64\ansys170.exe

Solver programs can potentially be launched from FEMAP using the *File, Analyze* command, the *Analyze* button in the *Analysis Set Manager*, the *Analyze Model* button on the *File* Toolbar, or the *Analyze* command on the context-sensitive menu for an Analyses in the *Model Info* tree.

The *Echo Command Line to Message Window* will simply echo the full path to the solver executable along with any command line arguments which FEMAP uses automatically along with any specified in the *Arguments* field.

Any FEMAP-specific solver environment variables currently specified on your system will automatically be used to populate the appropriate fields on the *Solvers* tab the first time you choose the *File, Preferences* command.

Note: In each section of the *Solvers* tab, the combination of the path specified in the *Program* field and any item(s) entered in the *Arguments* field is used to generate a “command line” which will be used to launch the corresponding solver. It is up to user to verify that this “command line” is valid, therefore, if the solver will not run when entering the same command line into a windows command prompt, it will also not run when launched with FEMAP.

In addition, the user MUST also verify the selected solver is appropriately licensed in order for the solver to run properly, as FEMAP is not responsible for licensing any “linked solver”.

Geometry/Model

- Added *Allow Solid Boolean to Create NonManifold Geometry* option to *Geometry Preferences* section.

When on, allows any “Boolean” command on the Geometry, Solid… menu (*Geometry, Solid, Add/Remove/Common/Embed/Intersect* commands) to potentially create NonManifold geometry (i.e., a Parasolid “General Body”) as a result of the operation. When off, if any operation would create a NonManifold body, you will be asked “Ok to allow this operation to result in a NonManifold Solid?”. Answering *Yes* will create a General Body, while answering *No* will cause the command to fail.

- Updated the *Element Quality Preferences* dialog box accessed via the *Element Quality* button by adding 3 new element quality checks to the *NX Nastran* tab (*Quad AR*, *Tria AR*, and *Tria EPLR*) and changing all operators on the *NX Nastran* tab from “<=” or “>=” to “<” or “>”, which matches the functionality of GEOMCHECK.

Results

- Added *Auto Upgrade Abaqus ODB Database* option to *File Options* section. When enabled, the Abaqus ODB Database will automatically be updated to the most current version of the ABAQUS ODB Database supported by that version of FEMAP.
- Added *Nastran Options* section and moved two existing options, *Output Set Titles* (formally *Nastran Output Set Titles*) and *Append Femap Title*, from the *File Options* section into this section.
- Added *Use Static Subcase IDs* option to *Nastran Options* section.

When this option is enabled, an attempt will be made to create Output Sets using IDs corresponding to results for specific Subcase IDs in a Nastran Output File which has been imported or attached. If an output set with that ID already exists in the FEMAP model, then the next empty ID after the Subcase ID will be used.

For example, if results exist for Subcases 100 and 200 in the Nastran Output File, and no Output Sets exist in the model, then corresponding Output Sets with IDs of 100 and 200 will be created. If this same Nastran Output File were to be imported again, then the newly created Output Sets would be given the next empty Output Set IDs, which for this example would be 101 for Subcase 100 and 201 for Subcase 200.

- Added *Track Revision* option to *Nastran Options* section.

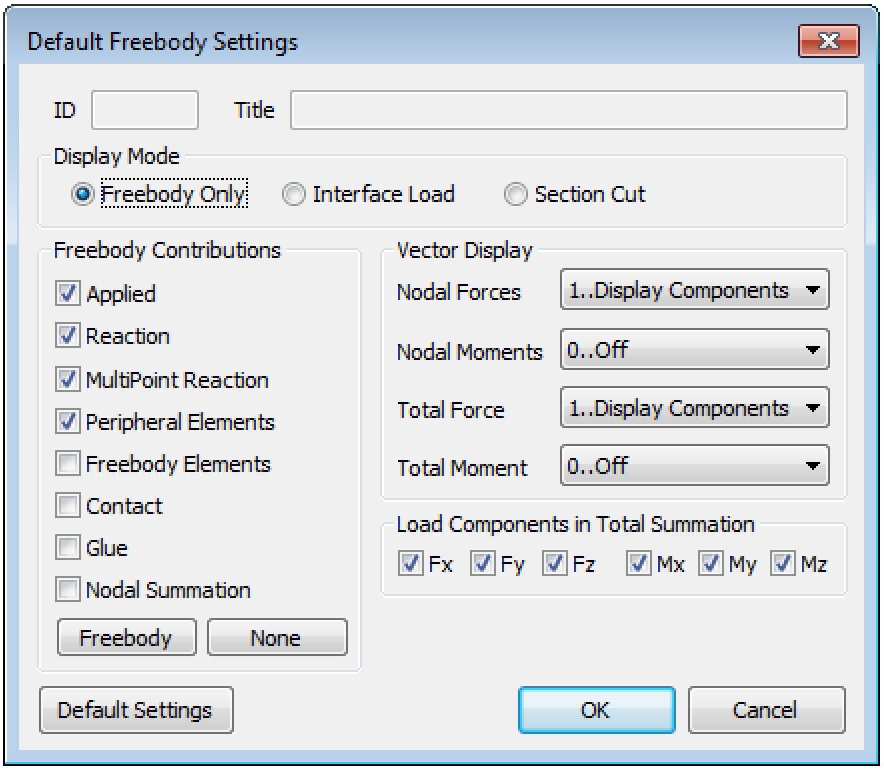
When this option is enabled, a Nastran Subcase ID will be stored as the “Case” ID on each Output Set created by importing or attaching to a Nastran Results File. If another Nastran Output File containing the same Nastran Subcase ID is imported or attached, the appropriate “Revision” number will also be stored on the Output Set.

For example, Output Set 1 (Case ID = 100, Revision = 0), Output Set 2 (Case ID = 200, Revision = 0), and Output Set 3 (Case ID = 100, Revision = 1) already exist in the model. If a Nastran Output File containing results for Case 100 and Case 200 is imported or attached, then Output Set 4 would be created with Case ID = 100, Revision = 2 and Output Set 5 would be created with Case ID = 200, Revision = 1.

Note: When this options is enabled, a “-C (Subcase ID)” will append each output set title. If the output set is a “Revision”, then a “-C (Subcase ID) -R (Revision #) will append each output set title.

- Added *Freebody Defaults* section and added the *Set Freebody Defaults* button.

The *Set Freebody Defaults* button in this section will open the *Default Freebody Settings* dialog box:



This dialog box allows you to specify the default settings to be used every time a new Freebody entity is created in the model. Default options may be specified for *Display Mode*, *Freebody Contributions*, *Vector Display*, and *Load Components in Total Summation*. Clicking the *Freebody* button in the *Freebody Contributions* section will select only *Applied*, *Reaction*, *MultiPoint Reaction*, and *Peripheral Elements*, while clicking the *None* button will turn off all items in the *Freebody Contributions* section. Clicking the *Default Settings* button will cause all settings to match the picture above.

Note: Typically, when *Display Mode* is set to *Freebody Only*, you are not able to select an option for *Total Force* or *Total Moment* or select components in the *Load Components in Total Summation* section. Because this dialog is used to set default values, they are available and will be used if the *Display Mode* of a newly created Freebody entity is later changed from *Freebody* to *Interface Load* or *Section Cut*.

For general information about using the Freebody Tool, see Section 7.2.3.3, “Freebody tool”.

Library/Startup

- Added *User Tools Path* field to *Startup Program File/Basic Script/Executable and Custom/User Tools* section.

FEMAP contains a toolbar called *Custom and User Tools*. This toolbar allows you to choose directories on your machine where you can access the “API scripts provided with the latest version of FEMAP” (*Custom Tools* defaults to the “API” directory shipped with FEMAP) and store any “custom commands and tools” created by you and/or an engineering organization (*User Tools*). Files used by *Custom Tools* and *User Tools* can be recorded Program Files (*.PRO or *.PRG files), FEMAP Basic scripts (usually *.BAS files), or “other” executable (for instance, a Visual Basic script compiled into a *.EXE file). The *Custom Tools* and *User Tools* icon menus on the *Custom and User Tools* toolbar will take any of those file types it locates in the specified directories and automatically place them into the appropriate menu structure found on the *Custom and User Tools* toolbar.

What’s New for version 11.2.1 and 11.2.2

User Interface

General

General

- Added dynamic face highlighting during Element Face picking and enabled dynamic highlighting of elements in the Connection Region dialog if “Elements-No Faces” is selected
- Added “Plane Element Offset” as an option when using the “Model Data Value” option for “Pick^” in the standard entity selection dialog box
- Added a Cancel button to the “OK to Start New Model (No=Add to Current Model)?” dialog box which appears when you “drag and drop” an analysis model or geometry file into an open instance of FEMAP
- Updated the “Add Connected Elements” option for “Pick^” in the standard entity selection dialog box to dramatically improve performance

Meshing

- Enhanced performance significantly when using the Mesh, Copy, Element; Mesh, Radial Copy, Element; Mesh, Scale, Element; Mesh, Rotate, Element; and Mesh, Reflect, Element commands to create copies of existing elements.

Enhanced performance significantly of the Mesh, Extrude commands to extrude elements, the Mesh, Revolve commands to revolve elements, and the Mesh, Sweep commands to sweep elements.

Aeroelasticity

- Added support for randomizing colors of the various Aero entities when modifying the color

Connections (Regions, Properties, and Connectors)

- Added the ability to copy Connections when using the Mesh, Copy...or Mesh, Rotate commands when the “Mesh Sizes, Loads, Constraints...” option is enabled and if both associated Connection Regions are also being copied (Added for 11.2.2)

Output and Post-Processing

- Updated “View, Advanced Post, Contour Model Data” command to allow display of “Plane Element Offset” on planar elements

Geometry Interfaces

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

FEMAP Interface	Latest Supported Version
Parasolid	Parasolid 28.0

- Enhanced support for importing geometry files from SolidWorks 2015 by implementing a newer version of the SolidWorks translator

Analysis Program Interfaces

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

- NX Nastran Interface
- DYNA Interface

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

NX Nastran Interface

- FEMAP with NX Nastran bundle now includes NX Nastran 10.2
- Added support for NX Nastran 10.2 Thermal and Mechanical Strain (op2 data blocks OSTR1ELC, OSTR1THC, OSTR1EL, OSTR1TH)
- Updated default value for the “Solid Results in Material CSys” option in the “NXSTRAT Solver Parameters” dialog box in the Analysis Set Manager to be Off instead of On, which will write a 0 to the ELRESCS field on the NXSTRAT entry instead of a 1, which is the default value for NX Nastran.

DYNA Interface

- Added “20..EQ 13: 1 point nodal pressure tetrahedron” as a formulation option for Solid elements to create ELFO=13
- Updated “Memory (Megawords)” field in the LS-DYNA Analysis Control dialog box, which writes the *KEYWORD MEMORY entry, to allow values up to 99,999,999, as the previous limit was 2,147
- Added “Max Matrix Strn DFAILM” and “Max T Shear Strn DFAILS” fields to Material Type “54..LS-DYNA Enhanced Composite Damage”, which write the DFAILM and DFAILS values for the *MAT_ENHANCED_-COMPOSITE_DAMAGE entry (Added for 11.2.2)
- Added “R Rot Constrained (1=Yes)”, “S Rot Constrained (1=Yes)”, “T Rot Constrained (1=Yes)”, “CST (0,1 or 2)”, and “SCOOR (-3 to 3)” fields to Material Type “71..LS-DYNA Cable Discrete Beam”, which write the RRCON, SRCON, TRCON, CST, and SCOOR values to the *MAT_CABLE_DISCRETE_BEAM entry (Added for 11.2.2)
- Added “CST (0,1 or 2)”, “SCOOR (-3 to 3)”, “R Rot Constrained (1=Yes)”, “S Rot Constrained (1=Yes)”, and “T Rot Constrained (1=Yes)” fields to Material Type “93..LS-DYNA Elastic 6DOf Spring Discrete Beam”, which write the CST, SCOOR, RRCON, SRCON, and TRCON values to the *MAT_ELAS-TIC_6DOF_SPRING_DISCRETE_BEAM entry (Added for 11.2.2)

OLE/COM API

New and modified API Objects and Attributes

- Added NumberOfLoads and NumberOfConstraints properties to the API Node object
- Added support for Boundary Surfaces in the API Entity Tracking object (Added for 11.2.2).

New and updated API Methods

- Added GetAllArray2 and GetCoordArray2 methods to Node Object (returns an extra Variant that indicates which nodes actually exist)
- Added GetAllArray2 method to Element Object (returns an extra Variant that indicates which elements actually exist)
- Updated the GetAllArray and GetCoordArray methods for the Node object to automatically zero all memory, so if you ask for entities that do not exist the values returned will all be zeroed
- Updated the GetAllArray method for the Element object to automatically zero all memory, so if you ask for entities that do not exist the values returned will all be zeroed

The following functions have been added or updated:

- feAppSetModel32 (method to supplement the existing feAppSetModel method for programming environments where 64-bit integers are not available)

- feAppGetModel32 (method to supplement the existing feAppGetModel method for programming environments where 64-bit integers are not available)
- Updated feSolidRemoveRedundantPoint to make it more useful when cleaning up imported CAD geometry where vertices that simply should not be relevant can now be removed (Updated in 11.2.2)

What's New for version 11.2

User Interface

General, Menu, Toolbars, Model Info tree, Meshing Toolbox, PostProcessing Toolbox, Charting pane, Entity Editor, Data Surface Editor, Entity Info, Data Table, API Programming

General

- Updated the dialog boxes which are used to select *Output Sets* and *Output Vectors* for a number of commands to have a toggle that allows the output sets to be shown in the list using Analysis Studies or simply as a list of all Output Sets. When shown using Analysis Studies, a toggle control exists at the top of each study to toggle on/off all Output Sets in the Analysis Study.
- Updated the *By Faces* and *By Output* pick methods for elements to follow the *Add/Remove/Exclude* setting and not always *Add*. Also, grayed the *Add Connected Elements*, *Add Connected Fillets*, and *Add Tangent Surfaces* methods if *Add/Remove/Exclude* is not on *Add*.
- Updated various *Define Element* dialog boxes to return the cursor focus back to the first field used to specify a node after changing the element topology in the dialog box. For example, the first of the *Nodes* fields will be active after changing from *Triangle* to *Quad* (or vice versa) in the *Define PLATE Element* dialog box.
- Updated the standard *Plane Locate* dialog box, when using the *Surface Normal* method, to no longer require you to specify a point for *At Point* or *Axis Point* if a planar surface has been selected for *On Surface*. If neither point is specified, the surface CG is used as the location for *At Point*.

Menu

- Added *Geometry, Surface, Extract* and *Geometry, Solid, Extend* commands to *Geometry* menu. Also, removed *Geometry, Solid, Slice Match* and *Geometry, Solid, Slice Along Face* commands, as this functionality has been consolidated in the enhanced *Solid, Geometry, Slice* command. See Geometry for more information.
- Added *Model, Output, Create/Manage Analysis Study* command to *Model* menu. See Output and Post-Processing for more information.
- Added *Mesh, Connect, Rigid* and *Mesh, Editing, Rigid Connectivity* commands to *Mesh* menu. See Meshing for more information.
- Added *Modify, Update Elements, Midusrface Thickness and Offset* command to *Modify* menu. See Element Update for more information.
- Added *List, Output, Freebody Section Cut* command to *List* menu. See Output and Post-Processing for more information.
- Added *Group, Operations, Generate Model Data Value* and *Group, Solid, Type* commands to *Group* menu. See Groups and Layers for more information.

Toolbars

- Added *Solid Slice with Plane* icon to *Solids* Toolbar. Also, added *Geometry Solid Slice Match* and *Geometry Solid Slice Along Face* commands to the *Additional Commands* section on the *Commands* tab of the *Customize* dialog box. These commands can be added to any toolbar or menu and provide access to the legacy *Geometry, Solid, Slice Match* and *Geometry, Solid, Slice Along Face* commands, respectively.
- Added *Smart Snap* as an option for the *Snap Mode* icon menu in *Select* Toolbar. Also, added *Hide Surfaces* and *Show Surfaces* commands to the *Visibility* submenu on the context-sensitive menu for *Solids.* Also, when using the *Dialog* command in the *Selector Actions* menu, the various picking methods available in the *Method^* menu are now available when a *Load* or *Constraint* entity type is set as the active entity type.
- Updated *Post Data* icon on *Post* Toolbar.

Model Info tree

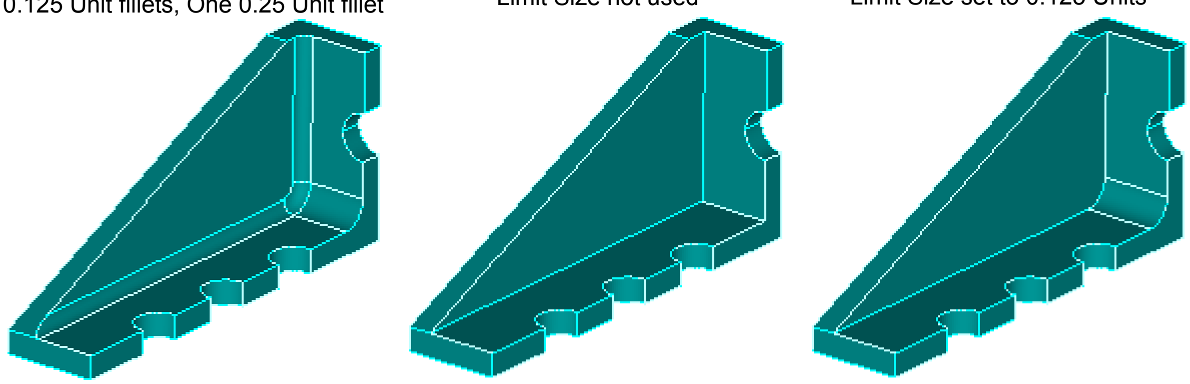
- Added *Hide Surfaces* and *Show Surfaces* commands to the “visibility check box” context-sensitive menu for *Geometry*. *Hide Surfaces* allows you to turn off visibility of the surfaces on the highlighted solids, sheet solids, or general bodies, while *Show Surfaces* will display any hidden surfaces again. If a body’s overall visibility is turned off, then the body is made visible again, the surfaces on the body will always also be visible again.
- Added *Analysis Study* and *All Results* branches under *Results*. The *Analysis Study* branch will contain all Analysis Studies currently in the model and any output set(s) currently residing in an analysis study will be listed under the appropriate Analysis Study. The *All Results* branch simply lists all output sets currently in the model.
- Added context-sensitive menu unique to the top-level *Results* branch. It contains *Attach to Results* (displays the *Manage Results Files* dialog box from the *File, Attach to Results* command), *Delete* (does the same thing as *Delete, Output, All* command), *No Deformation* (sets *Deformed Style* to *None - Model Only*), and *No Contour* (sets *Contour Style* to *None - Model Only*) commands.
- Added context-sensitive menu for Analysis Studies. It contains *Manage* (opens the *Analysis Study Manager*, same as *Model, Output, Create/Manage Analysis Study* command), *Copy*, *Edit, List*, *Renumber* (does the selected operation to any number of selected Analysis Studies), *Delete* (asks if you want to delete the output set(s) in the selected Analysis Studies along with the Analysis Studies), *Remove Study* (Deletes the selected Analysis Studies, but not the output set(s) in the Analysis Studies), *Animate Study* (sets *Deformed Style* to *Animate-MultiSet* and uses all output sets in the Analysis Study), and the *Envelope Study* menu (creates a *Max Value*, *Min Value*, or *Max Absolute Value* envelope of all output vectors in all output sets in the Analysis Study and also includes a *Create in Database* toggle).
- Added *Add to New Study*, *Add To Study*, and *Remove From Study* to context-sensitive menu for Output Sets. Also, added *Create in Database* toggle to the *Envelope* submenu on the context-sensitive menu for Output Sets.
- Added ability to move the vertical scroll bar up and down when a command dialog box is also open.

Meshing Toolbox

- *Feature Removal* tool - Added *Blends* to the *Feature Type* section. When used, prompts you to select an entire solid, sheet solid, or general body, instead of selecting individual blend (fillet) surfaces, then attempts to remove all blends. *Limit Size* restricts removal to blends with a radius less than or equal to specified value.

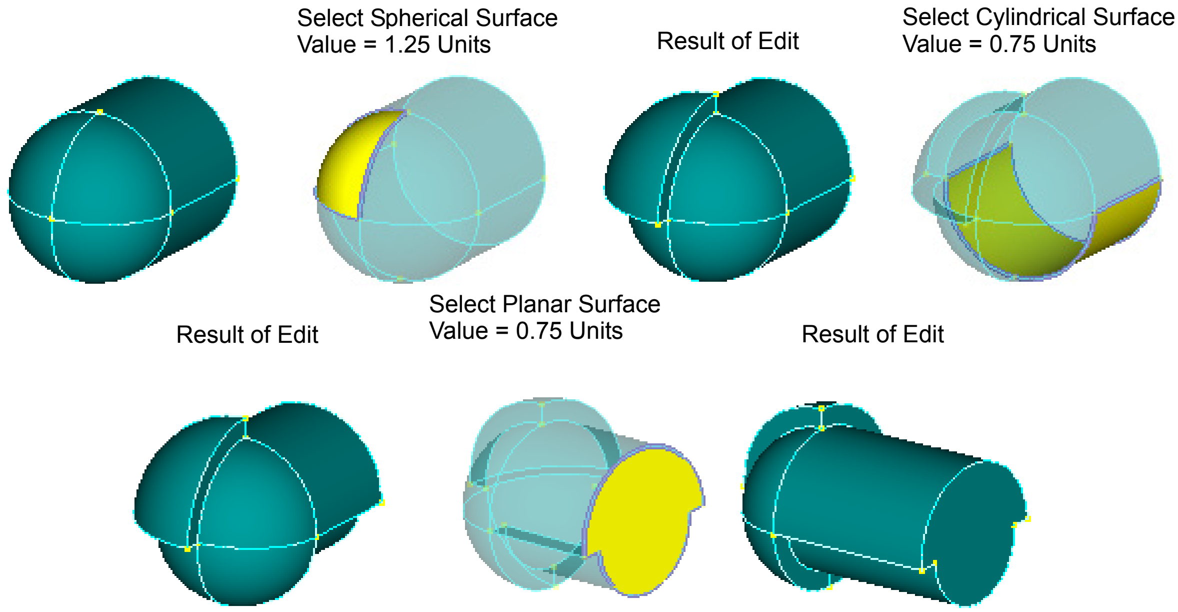
Here is an example of removing a “Blends” with and without using *Limit Size* enabled and set to 0.125 Units:

Original solid geometry with Two 0.125 Unit fillets, One 0.25 Unit fillet | Limit Size not used | Limit Size set to 0.125 Units

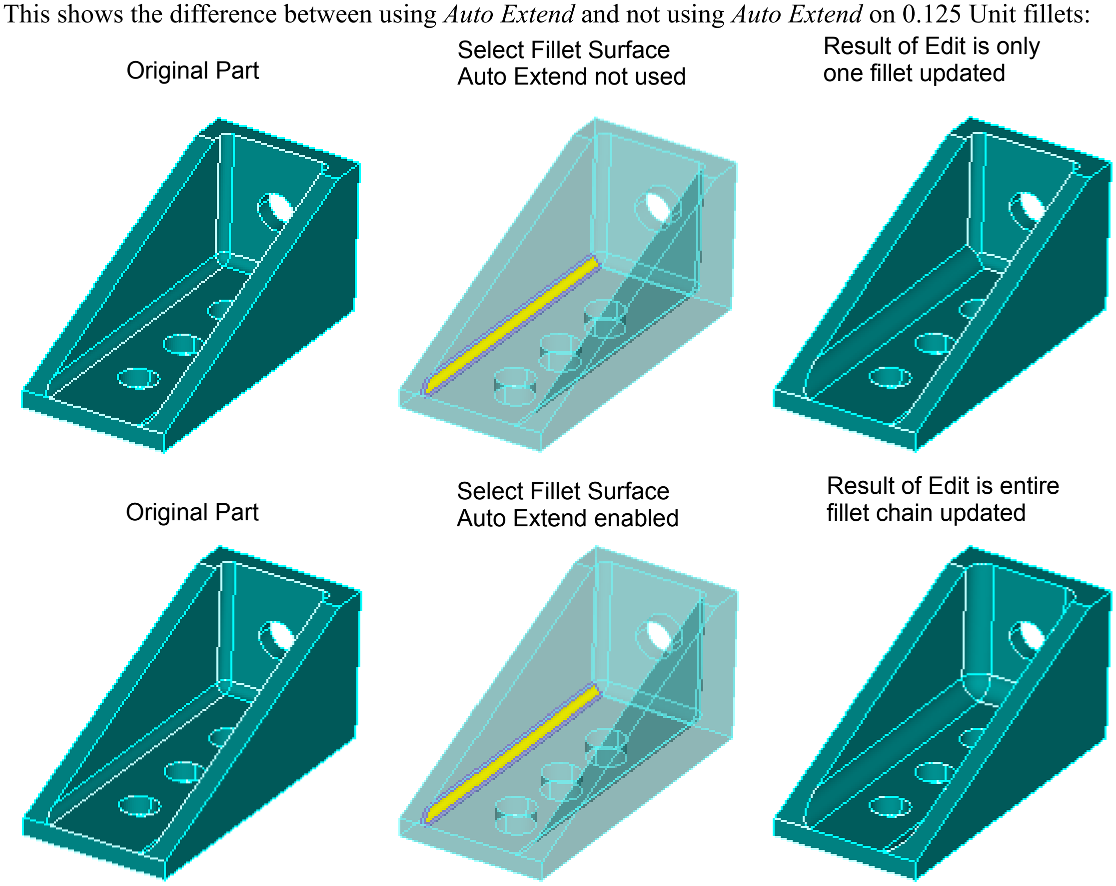


- *Feature Editing* tool - Added *Edit* to the *Operation* section. When used, this operation assigns a different size for cylindrical and spherical surfaces on a solid, sheet solid, or general body and is only available when *Selection Method* is set to *Surface*. Simply specify a *Value*, which effectively updates the radius of these types of surfaces, then select any number of surfaces. Typically, these types of surfaces are used for holes, fillets, and the outside of cylinders and spheres. If a planar or other non-cylindrical/non-spherical surface is selected, the *Value* indicates an offset distance with the direction determined by the surface’s normal. When enabled, *Auto Extend* will automatically select any additional surface(s) needed to update an entire “feature” (for example, both halves of a cylindrical hole, all surfaces of a fillet “chain”, or fillet surfaces connected to a planar surface). If 0.0 is entered for *Value*, you will be prompted to enter a value into a dialog box to use for the selected surface(s). If you enter 0.0 into the dialog box, then the command will attempt to remove the selected surface(s). If *Value* is negative, nothing will happen to cylindrical and spherical surfaces, but other types of surfaces will be offset that distance “inward” into the part.

In this example, both the spherical portion and cylindrical portion of the solid have a radius of 1.0 Units:



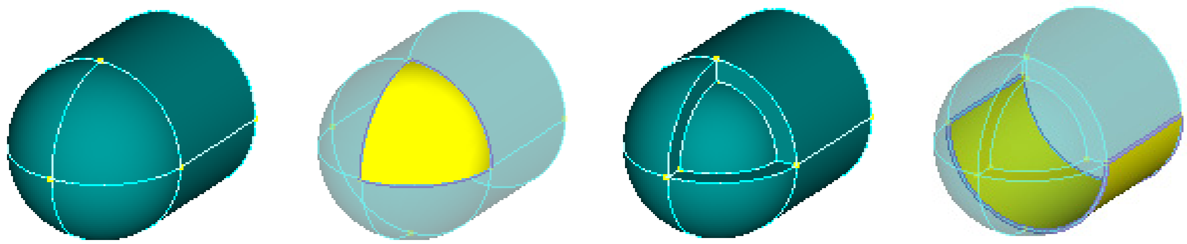
This shows the difference between using *Auto Extend* and not using *Auto Extend* on 0.125 Unit fillets:



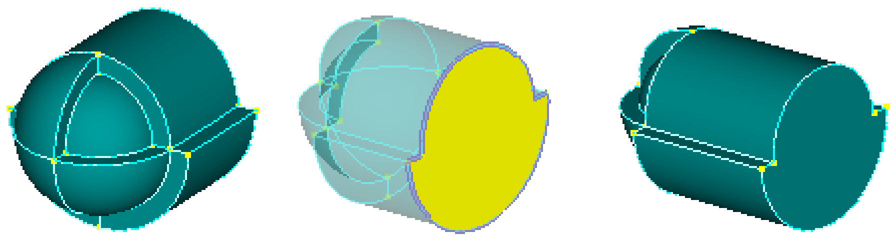
- *Feature Editing* tool - Added *Offset* to the *Operation* section. When used, this operation offsets surfaces on a solid, sheet solid, or general body and is only available when *Selection Method* is set to *Surface*. Simply specify a *Value*, which can be entered as positive value to offset “outward” from the body or a negative value to offset "inward" into the body. When enabled, *Auto Extend* will automatically select any additional surface(s) needed to update an entire "feature" (for example, both halves of a cylindrical hole or all tangent surfaces of an internal cavity/outer boundary). If 0.0 is entered for *Value*, you will be prompted to enter a value into a dialog box to use for the selected surface(s).

In this example, both the spherical portion and cylindrical portion of the solid have a radius of 1.0 Units:

Original Part
Select Spherical Surface Value = -0.25 Units
Result of Offset
Select Cylindrical Surface Value = 0.25 Units

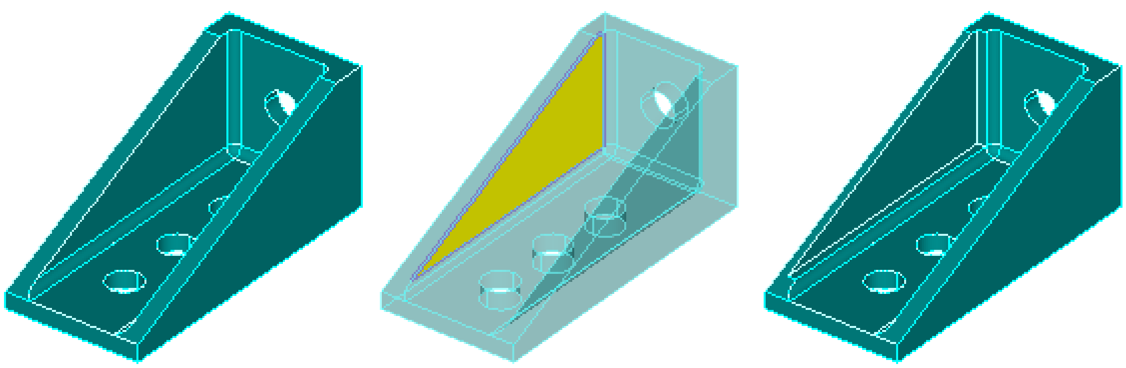


Result of Offset
Select Planar Surface Value = 0.75 Units
Result of Offset

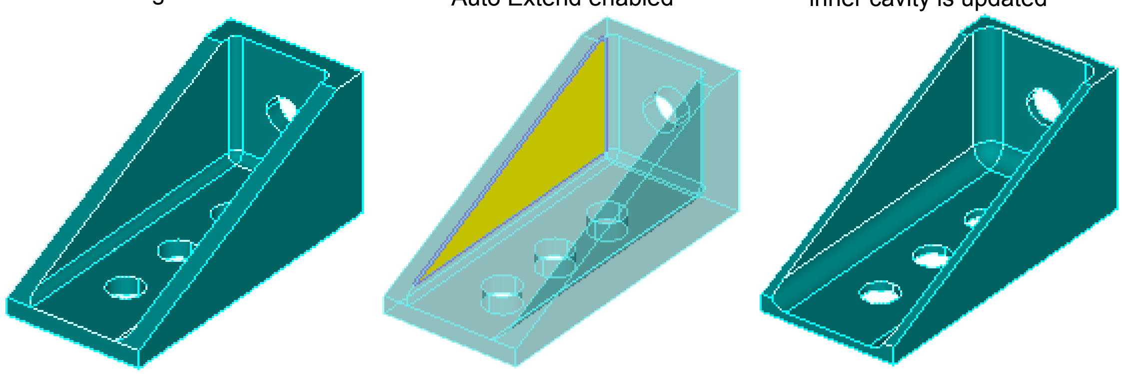


This shows the difference between using *Auto Extend* and not using *Auto Extend* using a value of -0.125 Units:

Original Part
Select Planar Surface Auto Extend not used
Result of Offset is only one surface updated

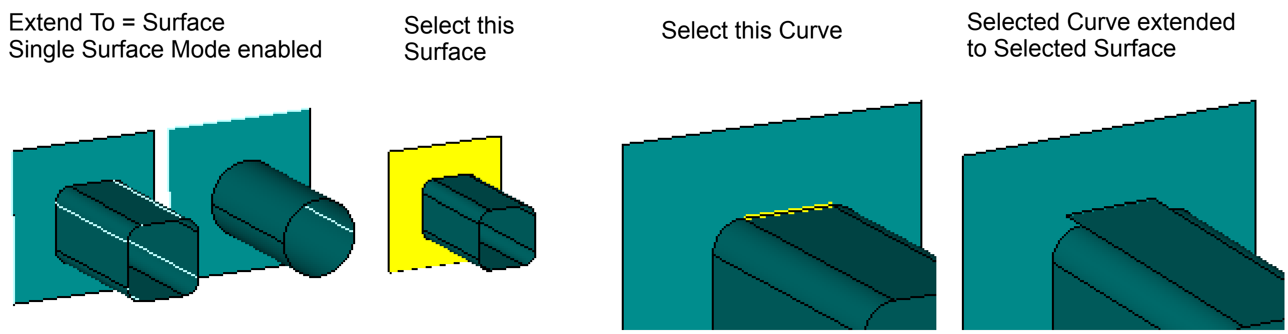


Original Part
Select Planar Surface Auto Extend enabled
Result of Offset is entire inner cavity is updated

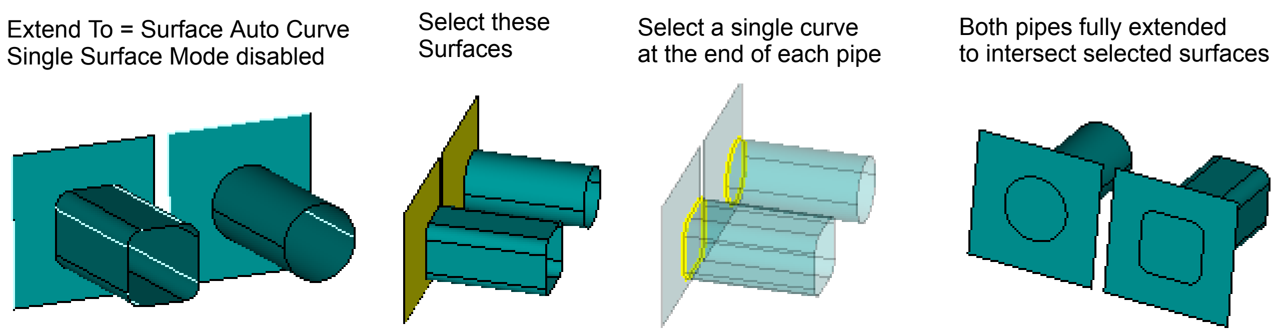


- *Geometry Editing* tool - *Operation* set to *Pad* or *Washer*, added ability to select either *Factor* or *Distance* in the *Offset Type* section to set the size of the “Pad” or “Washer”.
- *Geometry Editing* tool - *Operation* set to *Extend*, added *Surface* and *Surface Auto Curve* options to *Extend To* drop-down. Both *Surface* and *Surface Auto Curve* have two modes for selecting surfaces. When the *Single Surface Mode* is enabled, toggle on the *Entity Select* icon and you will be prompted to select a surface, then select a curve. Depending on which *Extend To* option you are using, that curve or curves will be extended to only that surface, then you will be prompted to select a surface and curve again. When *Single Surface Mode* is disabled, any number of surfaces on different solids, surfaces, or general bodies may be selected by clicking the “...” button located next to the “No Surface(s) Selected” button. Once selected, the number of selected surfaces will appear in the text of the button, which when clicked will highlight the selected surfaces. After surface selection is completed, choose any number of curves found on surfaces and they will be extended to the selected surface(s). At any time, use the *Clear Selection* button to reset surface selection.

Example showing how *Extend To* set to *Surface* can be used:

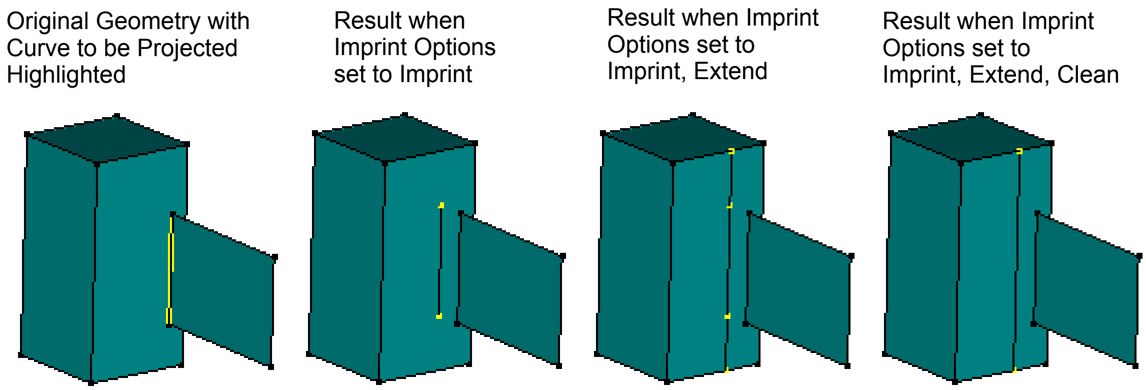


Example showing how *Extend To* set to *Surface Auto Curve* can be used



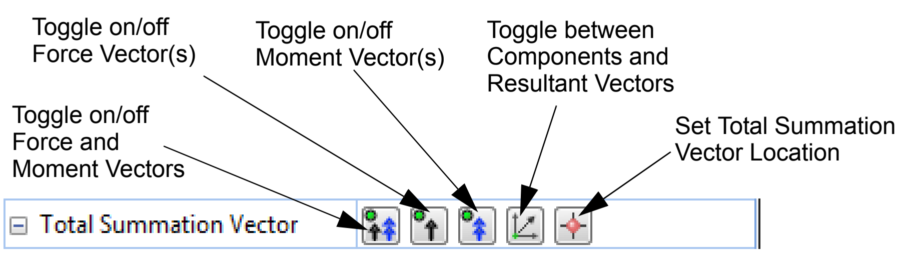
- *Geometry Editing* tool - Added *Project/Move Point* to the *Operation* section. When using *Project/Move To* set to *Solid, Surface*, or *Curve*, projects a point or points used by a surface, solid, sheet solid, or general body onto an entity of the selected type, thus modifying the original body. Turn on *Edge Aligned* to have the command attempt to follow the curvature of the edge connected to the point instead of simply projecting the point to the closest location on the target body. When using *Project/Move To* set to *Point*, it moves a point or points used by a surface, solid, sheet solid, or general body to the coordinates of the selected point, thus modifying the original body. To project multiple points at one time, use the *Dialog Select* icon after the point, curve, surface, or solid is selected using the *Entity Select* icon.
- *Geometry Editing* tool - Added *Project Curve* to the *Operation* section.When used, creates a new curve or curves on the selected surface using a normal projection. Using the *Entity Select* icon, you will prompted to select a surface, then a curve to project. To project multiple curves at one time, use the *Dialog Select* icon after the surface is selected. There are three different *Imprint Options* available. When using *Imprint*, the selected curve will simply be projected onto the selected surface, creating an imprinted curve. When using *Imprint, Extend*, the imprinted curve will be extended to the extents of the surface, but the points of the original imprinted curve will remain. When using *Imprint, Extend, Clean*, it simply adds a cleanup step to remove any redundant points, including the points on the original imprinted curve.

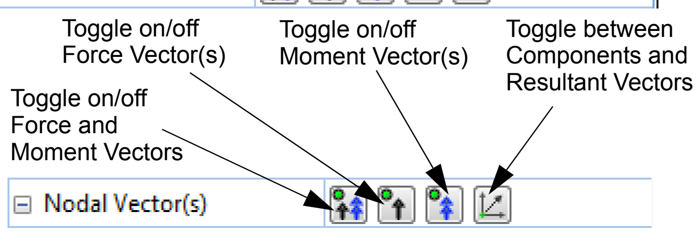
Here is an example of using the three different *Imprint Options*:



PostProcessing Toolbox

- Added *Select Post Data* icon to the toolbar, which displays the *Select PostProcessing Data* dialog box from the *View, Select* command.
- Added *Complex Results Mode* option to both the *Deform* and *Contour* tools, which allows you to set overall "Model Option(s)" or view-specific "override" values used to convert complex data to real data, on-the-fly, when post-processing complex results. See Output and Post-Processing for more information. Also, The *Complex Model Options* icon button can be used to set the "Model Option(s)" and the *Animation Phase Increment* from the active view will be used for *Synchronize Phase*, when *Style* in *Deform* tool is set to *Animate*.
- Added *Section Cut* as a *Display Mode* option for the *Freebody Tool*. Also, added a number of options which are used to control Freebody display when using a *Section Cut* Freebody. See Output and Post-Processing for more information.
- Added *Reverse Values* option under *Nodal Summation* in the *Freebody Contributions* section, which allows you to treat a solver-calculated nodal imbalance as a contribution. This can be useful when certain forces on a node, such as glue or contact forces, are not included in the grid point force table and result in an imbalance.
- For *Total Summation Vector* and *Nodal Vector(s)*, added icon buttons to toggle on/off display of "Forces and Moments", "Forces Only", or "Moments Only". Also, added an icon button to toggle between displaying "Component" vectors or a "Resultant" vector, along with an icon button to specify the location of the *Total Summation Vector*.





Charting pane

- Added *Relative to Node/Relative to Element* option to the *Vector vs. Entity* tab of the *Chart Data Series* dialog box. Only available when *X-Axis Values* is set to *Position*. When enabled, values be calculated relative to the X, Y, or Z location of the specified node for a nodal output vector or centroid of the specified element for an elemental output vector.

- Added ability to specify a *Color* for the “Major Grid Lines”. Also, added the *Dim for Dark Backgrounds* option, which selects a complementary color for the “Major Grid Lines” when using a darker background.

Entity Editor

- Added *Num Mesh Seeds* in the *Mesh Attributes* section for *Curves,* showing the number of “mesh seeds” (*Number of Elements*) on the curve currently in the *Entity Editor.*
- Added *Orientation CSys* for *Spring/Damper* elements, showing the Orientation Coordinate System for any Spring/Damper element referencing a CBUSH property currently in the *Entity Editor.*
- Added the *MSC Nastran Properties* section for *Connection Properties*, showing values on the *MSC Nastran* tab of the *Define Connection Property* dialog box for the Connection Property currently in the *Entity Editor.*

Data Surface Editor

- Added *Result Set Processing Data Surface* to *Create/Load Data Surface* icon menu. See Output and Post-Processing for more information.
- Added *Connection Manager Data Surface* to *Create/Load Data Surface* icon menu. See Connections (Region, Properties, and Connectors) for more information.

Entity Info

- Added *Orientation* for a Spring/Damper element which reference CBUSH Property. Depending on the method used for orientation, it will state “Node” with a node ID, “Vector” with XYZ coordinates, or “Coordinate System” with an Coordinate System ID. This information will also appear in the Tooltip for the element.

Data Table

- Added *Copy Rows - No Headers* command on context-sensitive for rows, which copies the content of the selected rows without including any of the column header information.
- Added *Add Nastran Element Checks* command to the *Show/Hide Columns* icon menu, which adds a column for each NX Nastran Element Quality Check available in FEMAP for each element currently in the *Data Table.*
- Added *Orientation CSys* and *CBush Property Orientation* columns for Spring/Damper elements being sent to the *Data Table.*
- Added *Num Mesh Seeds* column for *Curves* being sent to the *Data Table.*

API Programming

- Added *Save* icon to toolbar to allow you to simply save the file instead of always displaying a dialog box to perform a “save as” operation. If file has not been saved before a dialog box will appear to enter a file name.

Geometry

- Added *Geometry, Surface, Extract* command, which extracts the selected surface(s) from any solid, sheet solid, or general body in which the surface(s) currently reside. When removing surface(s) from a solid, the remaining surfaces of the solid will be changed from solid into a sheet solid or general body, whichever is more appropriate. If multiple connected surfaces are selected for extraction, then the extracted surfaces will be joined into a sheet solid or general body, whichever is needed.
- Added ability to use *Delete, Geometry, Surface* to delete individual surfaces from solids. Previously, you would have needed to use *Geometry, Solid, Explode* before these surfaces could be deleted.
- Added *Express* option to *Geometry, Surface, NonManifold Add* command. The *Express* option in *Processing Method* attempts to use a single operation in Parasolid to create a “General Body” from the selected surfaces, therefore performance can potentially be greatly improved.
- Added *Variable Thickness Processing* option to *Geometry, Midsurface, Automatic* command.

When on, *Variable Thickness Processing* will attempt to create midsurfaces at the “middle” of faces on solids which are not separated by a constant thickness. If an appropriate surface is created, it is often useful to then use the *Modify, Update Elements, Midsurface Thickness and Offset* command to create elements which follow the varying thicknesses in the part(s).

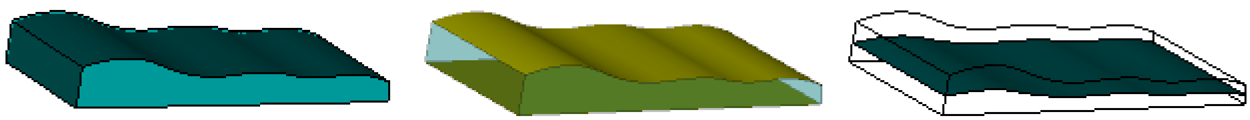
Note: If only two surfaces are selected on a solid, then an attempt will be made to create a midsurface between only those two surfaces. It may be helpful to use this process if selecting all of the surfaces on a solid did not produce an appropriate midsurface.

Here is an example:

Original Solid

Select these Surfaces, Variable Thickness Processing Enabled

Resulting Surface



- Added *Geometry, Solid, Extend* command, which allows you to extend a surface on a solid, sheet solid, or general body to a surface on another solid, sheet solid, or general body.

Typically, you would want the "extend from" body to at least partially intersect with the "extend to" body, but it is not required. In the case where no actual intersection between the bodies will take place, the selected surface on the "extend to" body will be "expanded" in all directions and, if possible, the surface of the "extend from" body will be extended to the "expanded" version of the surface on the "extend to" body.

Note: Regardless of the type(s) of bodies selected, the newly extended body and the "extend to" body will not be "added together" to form a single body after extension. If a single body is desired, simply use either *Geometry, Solid, Add* or *Geometry, Surface, NonManifold Add* at a later time to create a single body.

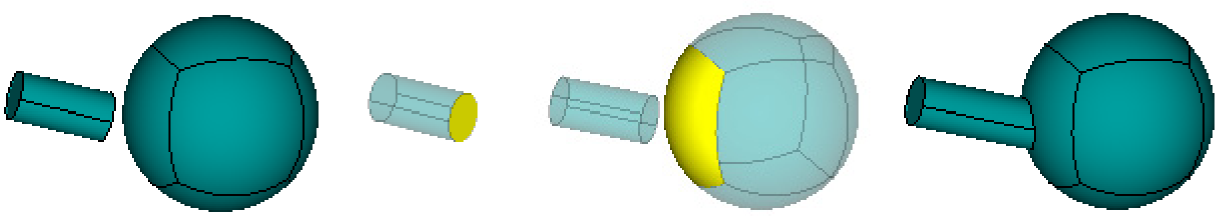
Here is an example:

Original Solids

Select this Surface on the From Body

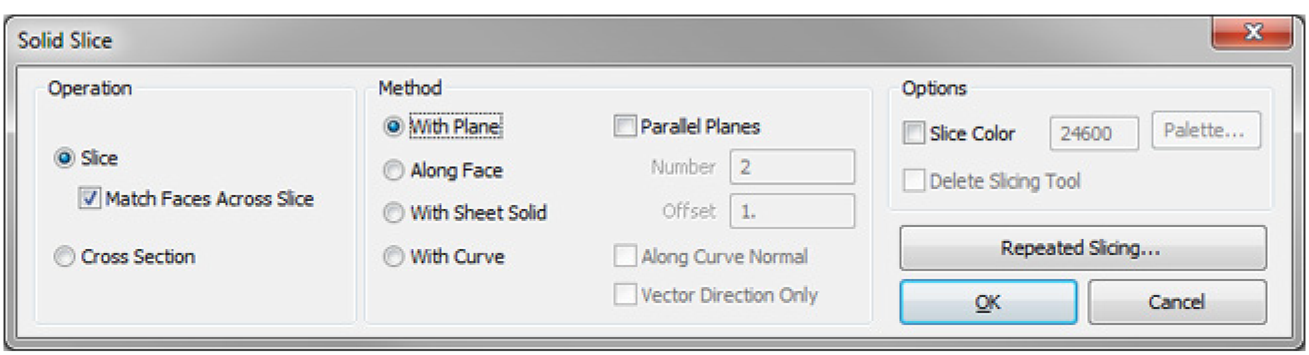
Select this Surface on the To Body

From Body extended to the To Body



- For Solids, added *Layer*, *Property*, *Type* (choose from *Solid*, *Sheet*, *Wire*, or *General/NonManifold*), *Using Curve*, and *Using Surface* options to the *Method^* menu in the standard entity selection dialog box.
- Updated the *Geometry, Solid, Slice* command to be a single command which consolidated the functionality from *Geometry, Solid, Slice*; *Geometry, Solid, Slice Match*; and *Geometry, Solid, Slice Along Face* commands.

This command begins by displaying the *Solid Slice* dialog box:



The *Solid Slice* dialog box allows you to select an *Operation*, either *Slice* or *Cross-Section*, to control what type of entities will be created when the command is completed. The dialog box also allows you to select a *Method* and some additional *Options* to control the type of entities used as the "slicing tool(s)". Once the *Operation, Method*, and *Options* have been specified, press *OK* or *Repeated Slicing* to display the standard entity selection box to the select the solid(s). Depending on the *Method*, a dialog box will then be displayed to specify a plane or select the appropriate slicing tool(s). Once changed, *Operation* and *Method* will “persist” in a session until changed again.

If you select *OK*, then the selection of solids, sheet solids, or general bodies will be followed by specification of a plane or selection of slicing tool(s), then the command will exit.

If you select *Repeated Slicing*, then the selection of solid(s) will occur only once, but the specification of a plane or selection of slicing tool(s) will repeat until you choose *Cancel*. Also, any additional solid(s) created during a single use of the command will also be sliced, when appropriate, with each plane or slicing tool.

Note: Three “legacy” slice commands, *Geometry Solid Slice With Plane*, *Geometry Solid Slice Match*, and *Geometry Solid Slice Along Face*, can be added to any menu or toolbar. To find them, use the *Tools, Toolbars, Customize* command, select the *Commands* tab, then choose *Additional Commands*.

Operation

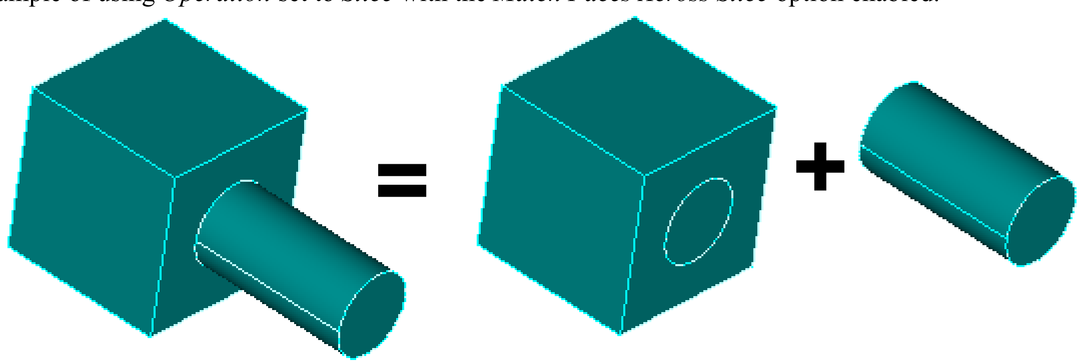
Specifies the type of *Operation* to be attempted by the command.

Slice - any selected solid will be divided into two or more solids, provided the specified plane intersects the solid(s) or selected “slicing tool(s)” pass through the entire solid.

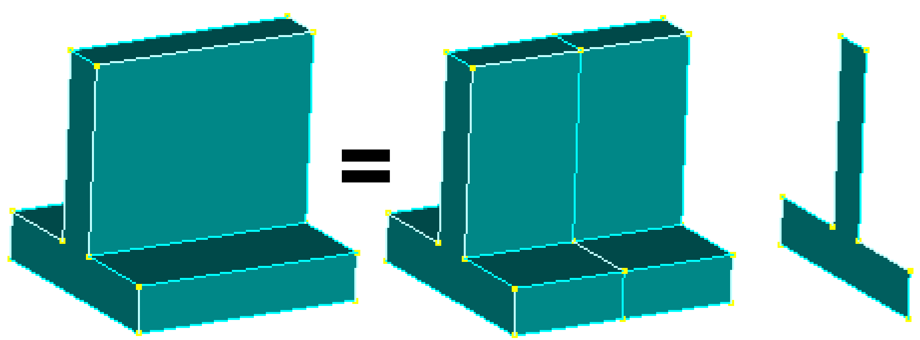
Match Faces Across Slice option - when enabled, an attempt will be made to create matching faces across the slice on both solids created from the *Slice* operation. See example below. Enabled/disabled persists in session.

Cross Section - individual surfaces will be created in the interior of each selected solid (i.e., a cross section), provided the specified plane intersects the solid(s) or selected “slicing tool(s)” pass through the entire solid. See example below.

Example of using *Operation* set to *Slice* with the *Match Faces Across Slice* option enabled.



Example of using *Operation* set to *Cross Section* using a plane at the midpoint of one of the curves:.



Original Solid | Original Solid and Cross Section Surface | Cross Section Surface Only

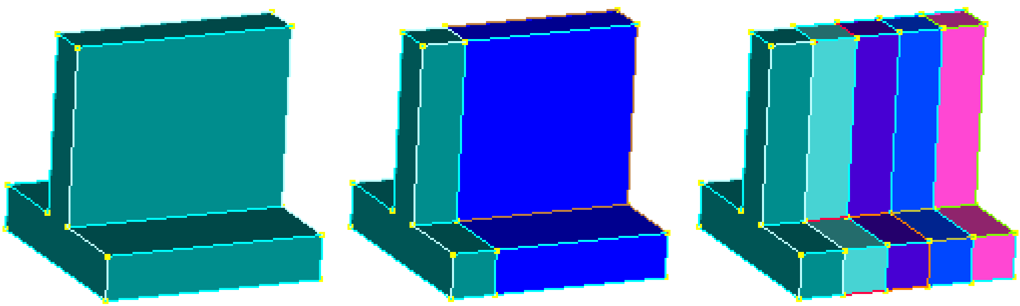
Method

Controls the *Method* used to slice the solid(s), which can either be a specified plane or selected “slicing tool(s)”.

With Plane - the selected slicing *Operation* will be attempt to use a plane specified using the standard *Plane Definition* dialog box. For more information on the various methods available for specifying a plane, see Section 4.3.4, "Plane Definition" in the FEMAP User Guide.

Parallel Planes option (only available when *Method* is set to *With Plane*) - when enabled, an attempt will be made to slice the selected solid(s) using a specified *Number* of additional planes which are parallel to the specified plane, with each plane separated by the *Offset* distance.

Example of using *Operation* set to *Slice* and *Method* set to *With Plane* showing a single plane and when using the *Parallel Planes* option with *Number* set to 4 (solids shown different colors for clarity):



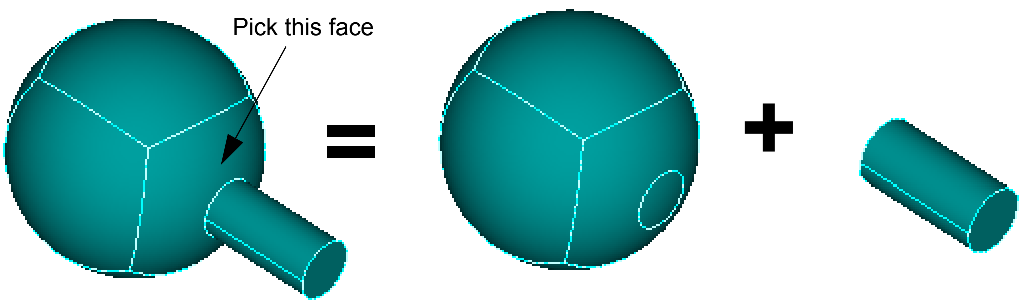
Original Solid

Original Solid sliced with Single Plane

Original Solid sliced with 4 Parallel Planes

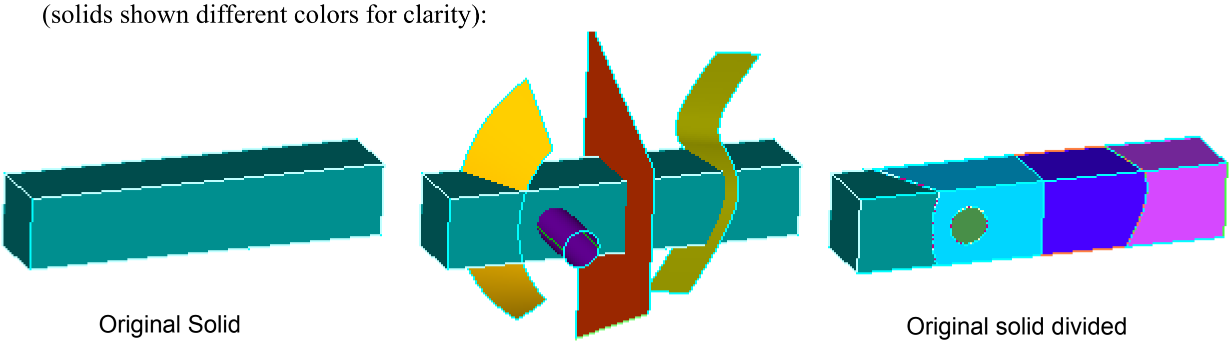
Along Face - a face of the solid is selected instead of specifying a plane. The face can be planar or curved.

Example of using *Operation* set to *Slice* and *Method* set to *Along Face* when selecting a curved surface:



With Sheet Solid - any number of sheet solids (surfaces) can be selected as “slicing tool(s)” to subdivide the selected solid(s). Each “slicing tool” must pass completely through at least one of the selected solid(s) in order to be used successfully. To delete “slicing tool(s)” after use, enable *Delete Slicing Tool* in the *Options* section.

Example of using *Operation* set to *Slice* and *Method* set to *With Sheet Solid* with *Delete Slicing Tool* option enabled (solids shown different colors for clarity):



Original Solid

Orange, Purple, Red, and Yellow Surfaces used as Slicing Tools

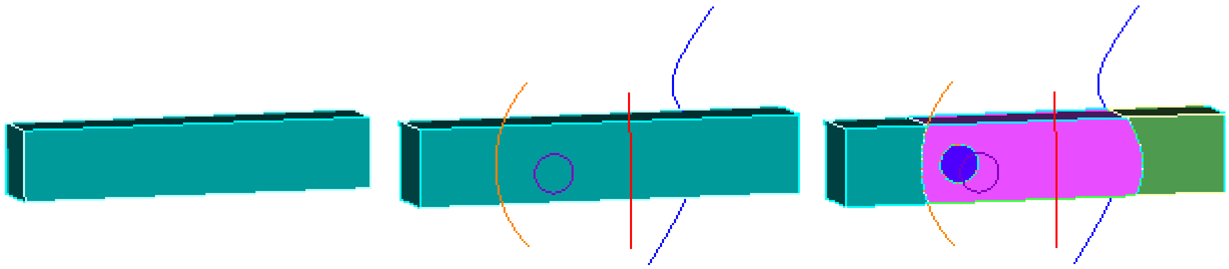
Original solid divided into 5 individual solids

With Curve - any number of curves can be selected as “slicing tool(s)” to subdivide the selected solid(s). Each “slicing tool” should be either as wide least one of the selected solid(s) or be a shape that can be used to “cut” a shape into the interior of one of the selected solids in order to be used successfully. To delete “slicing tool(s)” after use, enable *Delete Slicing Tool* in the *Options* section.

Along Curve Normal option (only available when *Method* is set to *With Curve*) - when enabled, any curve selected as a “slicing tool”, which has a normal direction that can be determined will be extended in both directions along that normal vector in an attempt to slice the selected solid(s). If a normal vector for any curve cannot be determined (i.e., non-planar curve) it will be ignored by the command.

Vector Direction Only option (only available when *Method* is set to *With Curve*) - when enabled, any curve selected as a "slicing tool" will only be extended in the specified vector direction in an attempt to slice the selected solid(s).

Example of using *Operation* set to *Slice* and *Method* set to *With Curve* with *Along Curve Normal* enabled (solids shown different colors for clarity):



Original Solid

Orange, Purple, Red, and Blue Curves used as Slicing Tools

Original solid divided into 4 individual solids (Red Curve ignored)

Options

Controls the color of surfaces created on solids due to the slicing Operation and if slicing tool(s) will be deleted.

Slice Color - when enabled, sets the color of any surface(s) created by a slicing operation to the selected color. Click the *Palette* button to select a color from the *Color Palette* dialog box. Available for either *Operation* and when using any *Method*.

Delete Slicing Tool - when enabled, deletes any Sheet Solid or Curve selected as a slicing tool after it is used for an Operation. Available for either *Operation*, but only when *Method* is set to *With Sheet Solid* or *With Curve*.

Hint: This command is extremely useful when importing CAD files of symmetrical parts. Most solid models in CAD systems will be of the entire model to generate drawings. You can use this command to slice the part through its plane(s) of symmetry and produce a much smaller and efficient model for meshing and analyzing. If you need to mesh the entire model due to non-symmetric loading conditions, simply mesh the sliced portion and then reflect the mesh. You will be able to produce a much better mesh in less time, than if you attempt to mesh the entire part. You will also be guaranteed to obtain a symmetrical mesh

Meshing

- Added *Mesh, Connect, Rigid* command, which is used to automatically create rigid or interpolation elements. Simply select any number of "source nodes", then the command will use criteria specified in the *Connect Rigid Options* dialog box to automatically determine appropriate "target nodes" for each "source node". After creating the new elements, the command will then ask if you would like to edit the elements before finishing the command.

Rigid Element Type

Specifies the type of element to create. When *Rigid Element Type* is set to *RBE2*, rigid elements will be created and only *Dependent DOF (Target)* can be specified in the *Rigid Element Options* section. When *Rigid Element Type* is set to *RBE3*, interpolation elements will be created and both *Independent DOF (Target)* and *Dependent DOF (Source)* can be specified in the *Rigid Element Options* section.

Target Node Selection and Rigid Element Options

Target Node Selection offers a choice between *Automatic* and *User Defined* options for selection of "target nodes", while the *Rigid Element Options* section controls degrees of freedom, maximum allowed distance between a "target node" and a "source node", and maximum number of "target nodes" allowed for each "source node".

When using *Automatic*, all the nodes in the model are considered for attachment to the selected "source nodes", but determining which target nodes actually get attached to which "source node" is done using the value(s) specified for *Max Nodes* and *Max Distance* in the *Rigid Element Options* section. The specified *Max Distance* value is used to create a list of nodes which are within that distance from each "source node", then, when *Limit number of* *mapped nodes* in enabled, the *Max Nodes* value limits the "target nodes" used for each "source node" to the "n" closest nodes.

When using *User Defined*, click the *Select Nodes* button beside *Nodes* to select nodes using the standard entity selection dialog box. The user may optionally limit the previous node selection to free-edges only by clicking the *Select Elements* button beside *Limit to Free Edges (Optional)* to select elements to use for a free-edge search using the standard entity selection dialog box. Once nodes and/or elements have been selected, the text on the buttons will change to be *"# of selected" Nodes* or *"# of selected" Elements*, respectively. The icon buttons to the right of these buttons can be used to highlight the selected nodes or selected elements in the graphics window, but only one entity type can be highlighted at a time.

While the *Max Distance* value is always used with *Automatic*, it can be turned off entirely when using *User Defined* by simply unchecking the option for *Limit number of mapped nodes*. When both the *Limit number of mapped nodes* and *Limit search distance* options are on, the "target nodes" attached to each "source node" are determined in a similar manner as when using *Automatic*, with the lone difference being only the nodes selected in the *Entity Selection* section are considered instead of all the nodes in the model.

Note: When *Limit number of mapped nodes* is turned off, you may encounter some unexpected results, such as a large number of nodes being included as "target nodes" because they are all within the specified *Max Distance* value. Similarly, when *Limit Search Distance* is turned off, and Limit number of mapped nodes is turned on, then nodes from farther than expected may be included as "target nodes" because each "source node" will be connected to the number of nodes specified for *Max Nodes*, regardless of distance.

When *Rigid Element Type* is set to *RBE2*, only the *Dependent DOF (Target)* (default is all Translation DOF) can be specified, while no DOF are specified for the "source nodes". When *Rigid Element Type* is set to *RBE3*, the DOF on the "target nodes" are specified using *Independent DOF (Target)* (default is all Translation DOF), while the DOFs used for the "source nodes" are selected using *Dependent DOF (Source)* (default is all six DOF).

The default value for *Max Nodes* is always 10, while the default value for *Max Distance* is 1/10 of the "diagonal" of an "invisible box" that completely encapsulates all of the "source nodes".

Preview Target Nodes option

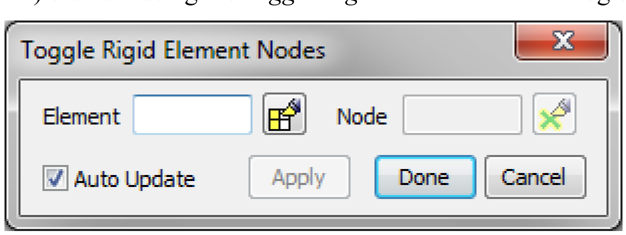
When the *Preview Target Nodes* is checked, all potential "target nodes" will be highlighted in the graphics window once *OK* has been pressed in the *Connect Rigid Options* dialog box. The standard entity selection dialog box will also be displayed to allow you to add or remove potential "target nodes."

In certain cases, some potential nodes may not be able to be paired with a source node. This may happen when a maximum distance has been specified and a "target node" is located too far from any "source node", or if a maximum number of pairings has been specified and no suitable "source node" is found that has less than the maximum number of pairings. When this happens, a message will appear asking "# highlighted target nodes were unable to be paired with a source node. Update the selector with these nodes?". If you choose *Yes* all of the unpaired nodes will be added to the *Selection List*, while if you click *No*, nothing with be done with the unpaired nodes.

Toggle Rigid Element Nodes dialog box

At this step in the process, rigid or interpolation elements have been created and a dialog box will appear asking "Edit rigid element connectivity?". Clicking *No* will simply end the command, while clicking *Yes* will display the *Toggle Rigid Element Nodes* dialog box, which works the same as the new *Mesh, Editing, Rigid Connectivity* command.

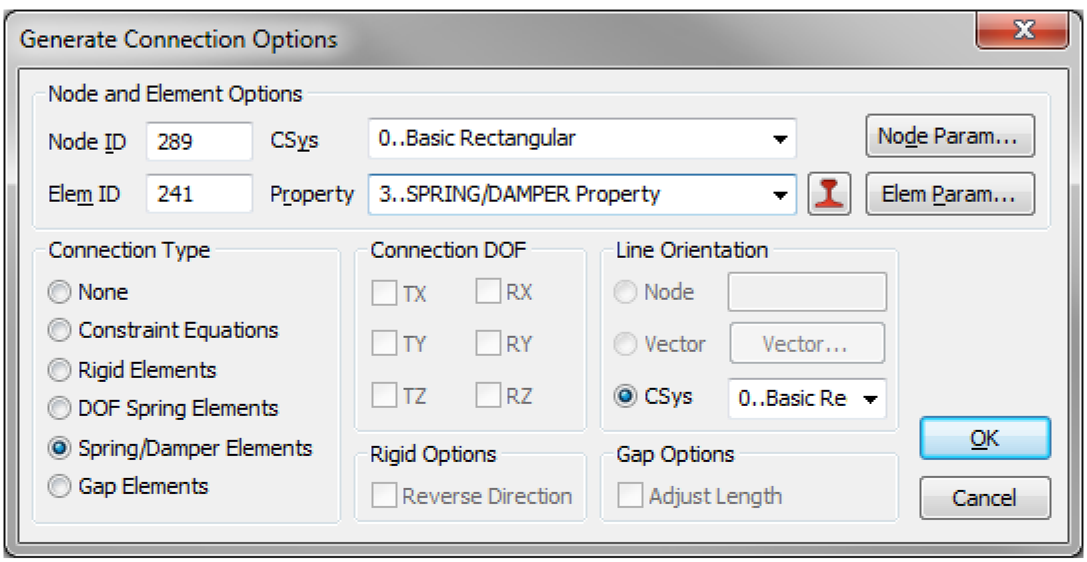
- Added *Mesh, Editing, Rigid Connectivity* command, which allows you to quickly edit an existing rigid (RBE2) or interpolation (RBE3) element using the *Toggle Rigid Element Node* dialog box:



To edit the *Dependent* nodes in a rigid element or the *Independent (Nodes To Average)* nodes in an interpolation element, simply select a rigid or interpolation element from the graphics window and it will highlight. When the *Auto Update* option is turned on, which is the default, selecting a node that is not currently used by the selected element will add that node to the element, while selecting a node which is already being used by the selected element will remove that node from the element. When the *Auto Update* option is off, you will need to select a node, then click the *Apply* button to add the node to the element or remove the node from the element. If a rigid element only has a single *Dependent* node or an interpolation element only has a single *Independent (Nodes to Average)* node, it cannot be removed from the element using this dialog box.

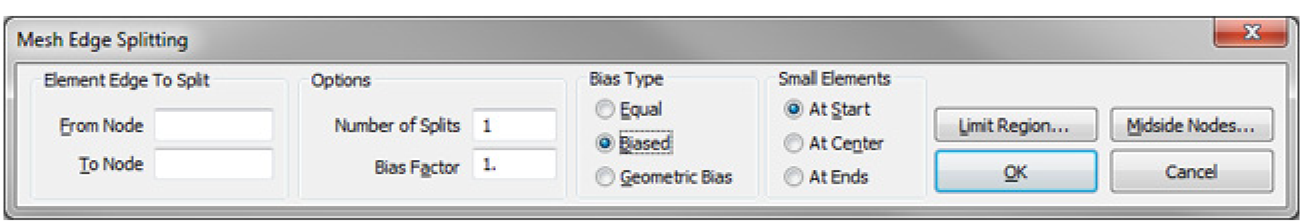
Note: The current DOF of the *Independent (Nodes to Average)* node with the lowest ID currently used by the selected interpolation element will be used as the DOF for any node added to the element.

- Updated the *Mesh, Geometry, Hex Mesh from Elements* command to allow you to specify *Mesh Spacing* options along with number of mesh *Layers* using the *Hex Layers Between Base and Top* dialog box. Choose from *Equal*, *Biased*, or *Geometric Bias*. When using *Biased* or *Geometric Bias*, specify a *Bias Factor* and choose a location for the *Small Elements* (*at Start*, *at End*, *at Center*, or *at Both Ends*). For instance, a *Bias Factor* of 2.0 and *Small Elements at Start* would produce a mesh where the layer of elements closest to the base region is 1/2 the thickness of the layer of elements closest to the top region.
- Updated the *Mesh, Geometry, Solids,* the *Mesh, Geometry, Solids from Surfaces*, and the *Mesh, Geometry, Solids from Elements* commands to allow you to select a value between 2 and 10 when using the *Multiple Tet thru Thickness* option. This option will create the selected number of tetrahedral elements through the thickness of a solid during tetrahedral meshing. Also, renamed the "Multiple Tet thru Thickness" option found in version 11.1 to *Split Tets With All Corners on Exterior* and moved it to the *Solid Automeshing Options* dialog box, accessed via the *Options* button in the *Automesh Solids* dialog box. Finally, removed the entire *Legacy Meshing Options* section (including all the options) from *Solid Automeshing Options* dialog box.
- Updated the first four commands on the *Mesh, Connect...* menu (*Closest Link*, *Multiple*, *Unzip*, and *Coincident Link*) by adding the *Line Orientation* section, which allows you to select an existing *Node*, specify a *Vector*, or select a *CSys* (Coordinate System, CBUSH elements only) to orient line elements created during the command.



- Updated the *Mesh, Editing, Edge Split* command to allow you to specify *Number of Splits*, a *Bias Type*, a *Bias Factor*, and a location for *Small Elements.*

This process in controlled using the Mesh Edge Splitting 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.

Options - Number of Splits

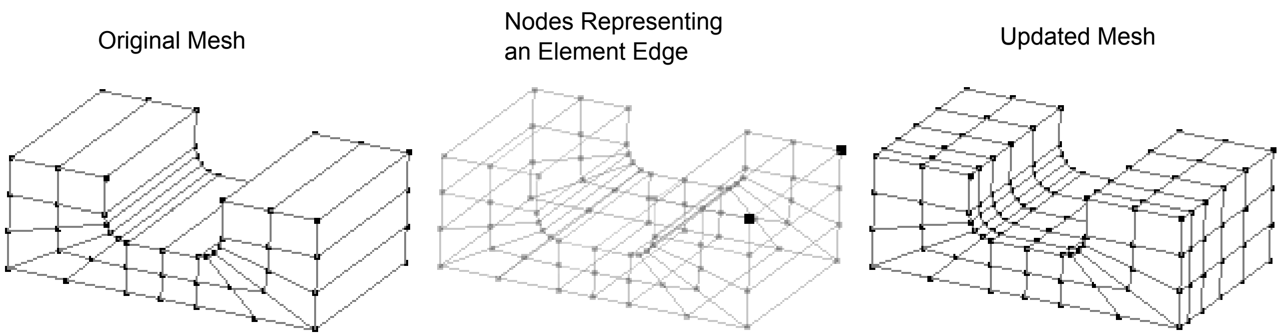
Number of Splits in the *Options* section must be an integer value. By default, has a value of 1 is used for *Number of Splits* per element, which simply splits the element down the middle. If *Number of Splits* is anything larger than 1, then the element will be split that many times. For instance, if Number of Splits is set to 3, each element will be split 3 times, causing there to be 4 elements for each original element.

Bias Type, Options - Bias Factor, and Small Elements

Optionally, a *Bias Type* can be chosen. If *Biased* or *Geometry Bias* is chosen, you must then enter a *Bias Factor* in the *Options* section in order for it to do any sort of biasing. The options in the Small Elements section control the location of the small elements when biasing. When using *At Start*, the elements will grow wider based on the *Bias Factor* as they move from the *From Node* to the *To Node*, so care should be taken in selecting the nodes in the appropriate order. When using *At Center*, the element will be thinnest at the middle of the original element and grow wider as they spread out from the center towards both the *From Node* and the *To Node*. Using *At Ends* is essentially the opposite of *At Center*, with the thinnest elements being closet to the *From Node* and *To Node* and the widest elements being at the center of the original element.

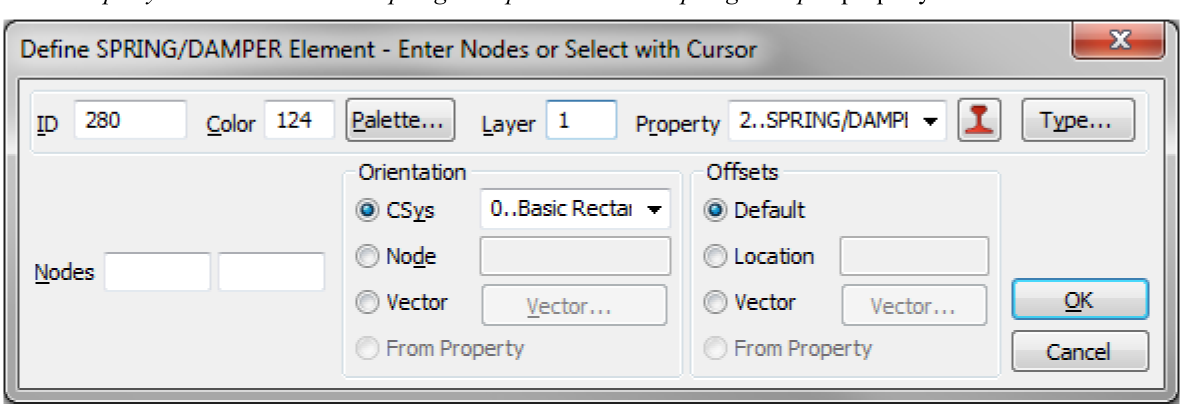
Any line, shell, or solid (Hex and Wedge only) element in the “split path” will be split, as long as the “split patterns” used in the splitting algorithm create elements with appropriate topology and maintain mesh connectivity.

Example of using *Number of Splits* set to 4 with *Bias Type* set to *Biased*, a *Bias Factor* of 3.0, and *Small Elements* set to *At Start*:



Elements

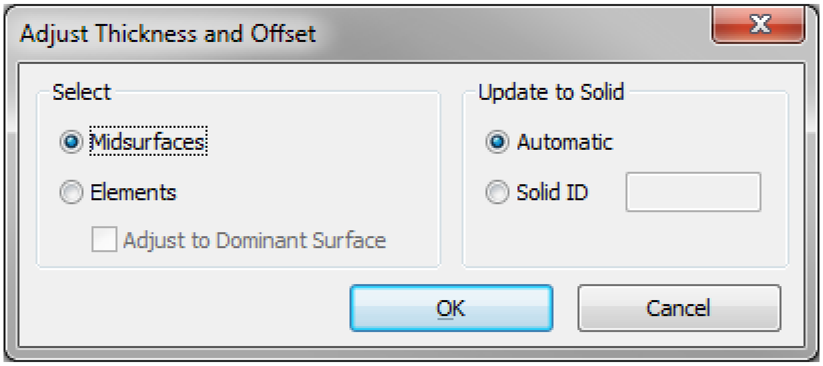
- Added ability to specify an “orientation coordinate system” and “location” for a *Spring/Damper* element referencing a *Spring/Damper* property with *Type* set to *CBUSH* on the element itself, not via the property. Simply set *Orientation* to *CSys*, then select an existing coordinate system from the drop-down control. The option to specify the orientation coordinate system on the property still exists by setting *Orientation* to *From Property*. Similarly, select *Location* in the *Offsets* section and enter a value to specify spring/damper location or use *From Property* to use the value for *Spring/Damper Loc* on the *Spring/Damper* property.



- Added ability to specify different thicknesses at each corner for *Plate* elements on the element itself, not only using the *T1*, *T2*, *T3*, and *T4* values on the *Plate* property. To set the corner thickness values on the element, use the *Modify, Update Element, Adjust Plate Thickness/Offset* command and choose *Element Thickness* in the *Update* section of the *Update Plate Element Thickness and Offset* dialog box instead of *Property Thickness*. The only way to edit these thicknesses is via *Modify, Update Element, Adjust Plate Thickness/Offset* command.

Element Update

- Added the *Modify, Update Elements, Midsurface Thickness and Offset* command to attempts to update the element thickness and apply appropriate offsets for elements to better match the original solid used to create the midsurface(s). Options for selecting the elements and the solids to use for the update are chosen in the *Adjust Thickness and Offset* dialog box:



The *Select* section allows you to specify if you will be selecting *Midsurfaces* themselves, which will then automatically select the plate elements associated with those midsurfaces, or any number of plates *Elements*. When the *Elements* is selected, there is an additional option to *Adjust to Dominant Surface*. When enabled, *Adjust to Dominant Surface* will adjust the element thickness and offsets of all selected elements based on the surface of each Solid used to determine the thickness and offsets for the largest number of selected elements.

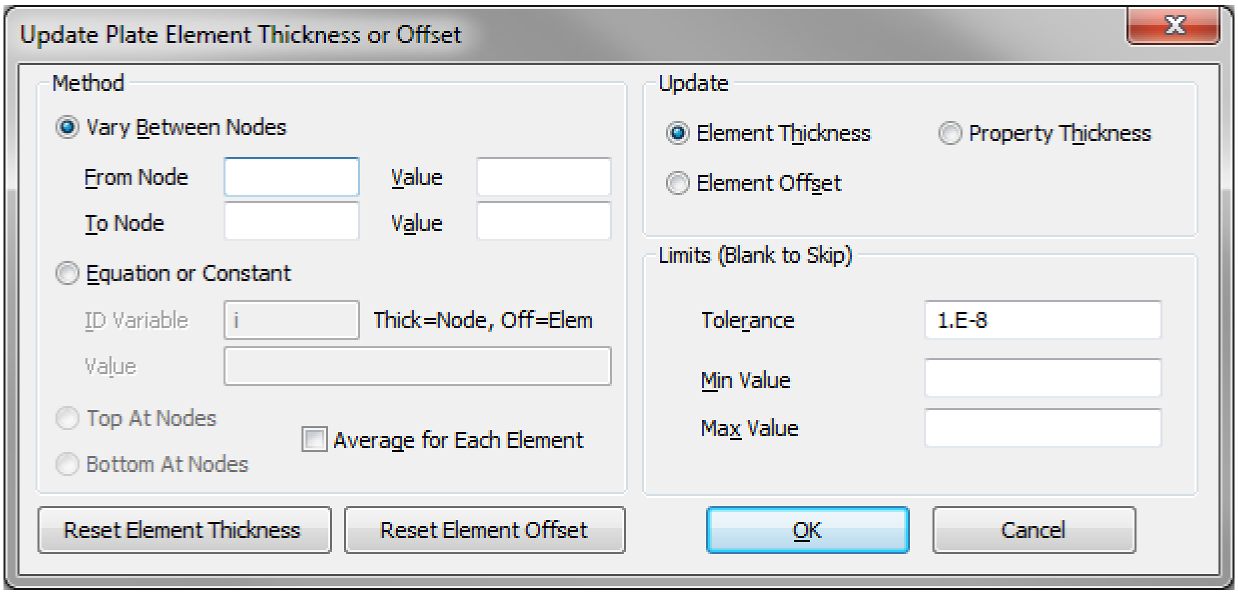
The *Update to Solid* section allows you to select how the solids used to determine thickness and offsets will be selected. When set to *Automatic*, each misdurface or element will automatically determine which solid to use, while when set to *Solid ID*, you will need to specify the ID of a solid or select s solid from the graphics window.

Selection of midsurfaces or elements is then done using the standard entity selection dialog box.

Hint: If selecting the *Midsurfaces* option does not produce the desired result, it may be a good practice to undo the command (*Tools, Undo* or Ctrl+Z), then try it again using the *Elements* option instead. This can also be said for trying both the *Automatic* and *Solid ID* options to see which yields the best results.

- Updated the *Modify, Update Elements, Line Element Orientation* command by adding the *Orientation CSys* option to allow you to select an existing coordinate system from a drop-down control (can only be used for *Spring/Damper* elements referencing a *Spring/Damper* property with *Type* set to *CBUSH*). Also, updated the *Radial* option. If you select a rectangular or cylindrical coordinate system, the orientation will be updated using a vector extending from the Z-axis of the selected coordinate system to the first node of each element. If you choose a spherical coordinate system, the orientation will be updated using a vector extending from the origin of the selected coordinate system to the first node of each element.
- Updated the *Modify, Update Elements, Material Orientation* command by adding the *-X*, *-Y*, and *-Z* options for *Coordinate Axis* in the *Material Angle* section, which allow you to use the opposite direction of the selected coordinate system axis to define material angle.
- Updated the *Modify, Update Elements, Orient Plate Normal/First Edge* command by adding four additional options to the *Normal* section. The *Away From Location* and *Toward Location* options require you to specify a location in space, then the normals will be adjusted so they will point away from or toward the specified location. The *Align to Vector* option will attempt to align the normals of the selected elements to match a specified vector, while *Align to CSys Direction* option will attempt to align the normals of the selected elements to match the selected axis and coordinate system specified in the *Options* section. Use the *Negative* option to align to the opposite direction of the selected coordinate system axis.

- Updated the *Modify, Update Elements, Adjust Plate Thickness/Offset* command by adding the *Element Thickness* option to the Update section to specify corner thickness values on the element itself. Also, added the Top At Face and Bottom At Face options to the offset elements so the top face or bottom face is aligned with the nodes. Also, added the *Reset Element Thickness* button to have selected elements revert to the thickness values on the Plate property and the *Reset Element Offset* button to set the offset values on all selected elements to 0.0.



- Updated the *Modify, Update Elements, Linear/Parabolic Order* command by adding the *Delete Midside Nodes* option, which is turned on by default when *Update To* is set to *Linear*. When on, any midside node not used by another element will be deleted. If a midside node is attached to an element which cannot have midside nodes, such as a rigid or interpolation element, then the midside node will be removed from the element where is was a midside node, but remain in the model connected to the element where it was not a midside node.

Properties

- Added *Surface with Reference Point* button which is available when *Shape* is set to *General Section* in the *Cross Section Definition* dialog box. The *Cross Section Definition* dialog box is accessed by clicking the *Shape* button in the *Define Property* dialog box when creating or editing *Bar*, *Beam*, or *Curved Beam* properties. If you push the *Surface with Reference Point* button, you will be asked to specify a coordinate to use for the *Reference Point* after selecting a surface and specifying a vector, and the *Reference Point* option will also be automatically enabled.

Aeroelasticity

- Added *PARAM OPPHIPA* option to *NASTRAN Aerodynamic Data (AEROx, MKAEROx)* dialog box, which when checked writes out PARAM,OPPHIPA,1 and will output the real vibration modes at all degrees of freedom, including the aerodynamic degrees of freedom.

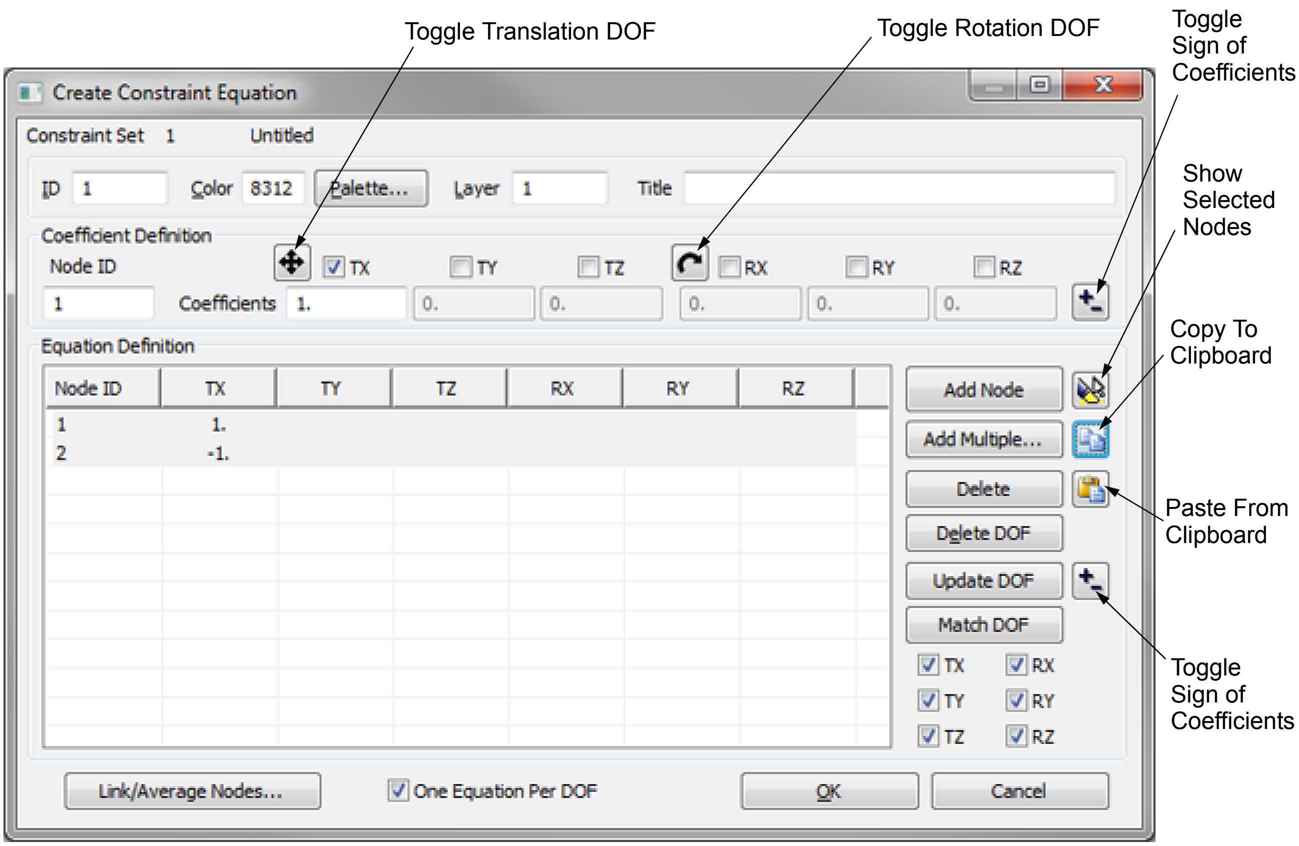
Loads and Constraints

- Added ability to create ACCEL1 entries for Nastran by creating *Acceleration* loads on nodes or geometry, then exporting a model from the *Analysis Set Manager* using an *Analysis Set* with *Analysis Type* set to “1..Static”.
- Added ability to *Model, Load, Nodal on Face* to apply a “Total Load” for *Force*, *Moment*, or *Heat Flux* loads. When enabled, which is the default, the “Total Load” option applies loads based on the face area, much like a geometry-based load. For parabolic faces, the load is “expanded” using the values and options specified for *Load Expansion on Midside Nodes* on the *Geometry/Model* tab of *File, Preferences.*
- Updated the *Model, Load, From Freebody* command, when using *Multi-Model* mode, by adding a *Closest Node* option and *Max Distance* field to the *Method* section of the *Create Load(s) from Freebody* dialog box. The *Closest Node* method essentially behaves the same as the *Match ID* method, but instead of searching the target

model for a node with a matching ID, which could be located anywhere, it searches within the specified *Max Distance* for the “closest node” in the target model. If a node is not within *Max Distance*, then no load will be created in the target model.

- Updated *Constraint Equations* to allow up to 6,000 terms (nodal DOF). Previously, the limit was 70 terms.
- Updated the *Model, Constraint, Equation* command to use a completely new *Create Constraint Equation* dialog box, which offers a number new options to improve the creation of constraint equations.

Equation coefficients are directly specified in the *Create Constraint Equation* dialog box:



ID, Color, Layer, and Title

These fields allow you to specify an *ID*, select a *Color*, specify a *Layer*, and/or provide a constraint definition *Title*.

Coefficient Definition

The *Node ID* field is used to specify the ID of an existing node. The *TX*, *TY*, and *TZ* check boxes can be used to turn on individual translation Degrees of Freedom (DOF) or click the *Toggle Translation DOF* icon button to turn on/off *TX*, *TY*, and *TZ*. Similarly, The *RX*, *RY*, and *RZ* check boxes can be used to turn on individual rotation DOF or click the *Toggle Rotation DOF* icon button to turn on/off *RX*, *RY*, and *RZ*. When a particular DOF is turned on, the field below becomes available for you to enter a coefficient for that DOF. The *Toggle Sign of Coefficients* icon button in this section will toggle the sign of the value in each of the active *Coefficients* fields (i.e., positive values will become negative values and vice versa). As always, the nodal degrees of freedom are in the X, Y and Z directions defined by the nodal output coordinate systems.

Add Node

Once a *Node ID* and any combination of DOF and *Coefficients* has been defined, click the Add Node button to add a row to the table in the *Equation Definition* section. If the node ID already exists in the table, you will be asked “OK to Update Coefficients for Node (ID of node)?”, answer *Yes* to update the DOF and coefficients in the table or *No* to not change anything.

Note: If you have at least one DOF enabled which has a coefficient defined other than 0.0, then selecting a single node from the graphics window will add that node and the specified coefficient(s) to the table.

Add Multiple

To apply the current DOF and *Coefficients* to any number of nodes, click the *Add Multiple* button and select the nodes using the standard entity selection dialog box. When you click OK, all of the nodes will be added to the table in the *Equation Definition* section with the same DOF and Coefficients.

Delete

Highlight any number of rows in the *Equation Definition* table (hold down *Ctrl* to choose multiple rows or hold down *Shift* to choose a range of rows), then click *Delete* to remove the rows from the table.

Delete DOF, Update DOF, Match DOF, Toggle Sign of Coefficients, and DOF check boxes

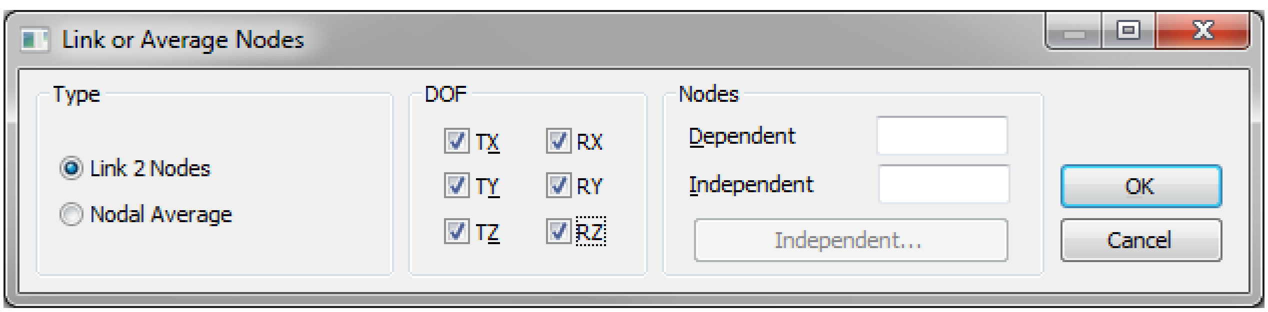
The *Delete DOF*, *Update DOF*, and *Match DOF* buttons only operate on the DOF that are currently checked, so if *TX* and *TZ* are checked, only those DOFs and/or coefficients will be changed by these buttons. Use *Delete DOF* to remove the coefficients in the column corresponding with the selected DOF from the highlighted rows. Use *Update DOF* to update the coefficients in the column corresponding with the selected DOF in the highlighted rows. The *Match DOF* button can only be used for rows which have a value defined in a single column and will copy that value into additional columns based on the currently selected DOF. The *Toggle Sign of Coefficients* icon button will update the sign of the coefficients in the column corresponding with the selected DOF in the highlighted rows.

Show Selected Nodes, Copy, and Paste icon buttons

The *Show Selected Nodes* icon button will highlight nodes represented by the rows highlighted in the table using the current settings for the *Window, Show Entities* command (See Section 6.3.2.3, "Window, Show Entities..."). The *Copy* icon button will copy the entire table to the clipboard. The *Paste* icon button will paste values created in another program, such as Excel, into the *Equation Definition* table.

Link/Average Nodes... button

Opens the *Link or Average Nodes* dialog box:



Depending on the option selected in the *Type* section, different controls become available.

When using *Link 2 Nodes*, select the desired DOF using the check boxes in the *DOF* section, then enter a node ID or select a node from the graphics window for both *Dependent* and *Independent*. When you click OK, the *Dependent* node will appear in the table with coefficient values of 1.0 for the selected DOF, while the *Independent* node will appear in the table with coefficient values of -1.0 for the selected DOF.

When using *Nodal Average*, select the desired DOF using the check boxes in the *DOF* section. First, enter a node ID or select a node from the graphics window for *Dependent*, which is typically the middle of a "spider", then click the Independent button to select the nodes "to be averaged" using the standard entity selection dialog box. When you click OK, the *Dependent* node will appear in the table with coefficient values of 1.0 for the selected DOF, while the *Independent* node(s) will appear in the table with coefficient values of "-1.0/number of selected nodes" for the selected DOF. For example, if 8 *Independent* nodes are selected, each will be added to the table with a value of -0.125 for each selected DOF.

Note: Any rows currently defined in the *Equation Definition* table will be deleted after using *Link/Average Nodes...* and pressing *OK* in the *Link or Average Nodes* dialog box. Additional rows can then be added.

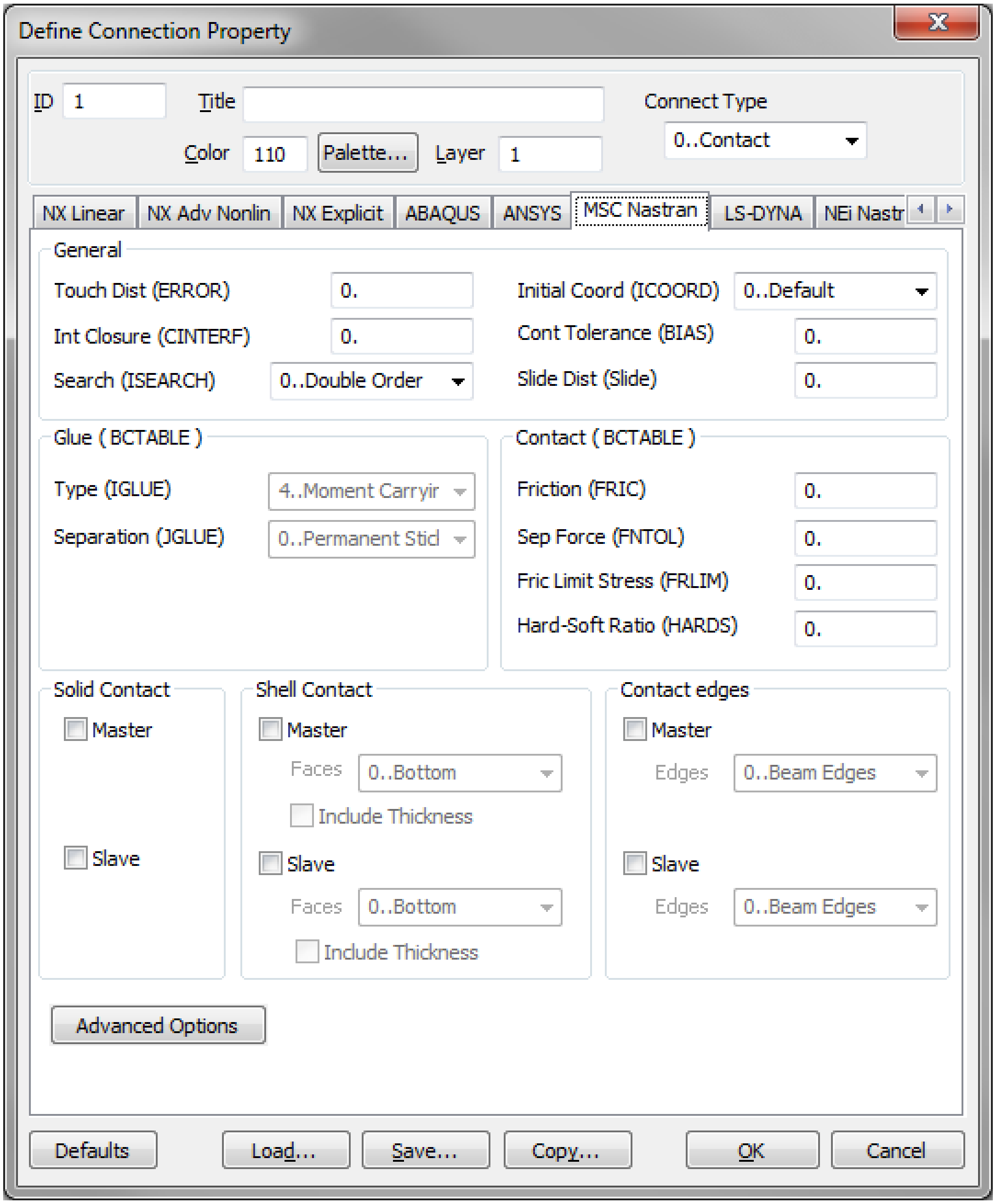
One Equation Per DOF

When this option is enabled, an individual constraint equation will be created for each DOF column which has coefficients defined. For example, if Node 1 has coefficient values of 1.0 for TX, TY, and RZ and Node 2 has coefficient values of -1.0 for TX, TY, and RZ, then 3 separate constraint equations will be created. If this option is disabled, then it creates a single constraint equation entity containing all coefficients for all DOF. In both cases, all constraint equations created by a single use of the dialog box will be placed into the same Constraint Definition.

Connections (Region, Properties, and Connectors)

- Added *Elements - No Faces* to the *Output* section of the *Connection Region* dialog box to support contact for MSC Nastran, which does not require elements faces. Also, added *MSC Nastran* section with *Friction* option to the *Connection Region Options* dialog box accessed by the *Region Options* button.
- Added *MSC Nastran* to *Define Connection Property* dialog box.

Pick the *MSC Nastran* tab to specify parameters found on the BCTABLE entry written for MSC Nastran.



The name of the parameter which will be written to a specific field in the BCTABLE can be found in parentheses after the name of the option (i.e., *Search (ISEARCH)* writes a value to the ISEARCH field of the BCTABLE entry).

General

This section contains inputs which are used for both Linear Contact (SOL 101 Only) and Glued Contact (all supported solution sequences, SOL 400 and 600 are not currently supported). The values for *Touch Dist (ERROR)*, *Int*

Closure (CINTERF), *Search (ISEARCH)*, and *Initial Coord (ICOORD)* are written to the appropriate fields on the first 2 "SLAVE" lines of the BCTABLE entry, while the values for *Cont Tolerance (BIAS)* and *Slide Dist (SLIDE)*, when non-zero, are written to the "FBSH" line.

Glue (BCTABLE)

This section is only available when *Connect Type* is set to "1..Glued". The selected option for *Type (IGLUE)* is written to the first "SLAVE" line of the BCTABLE entry, while the option selected for *Separation (JGLUE)* is written to the second "SLAVE" line.

Contact (BCTABLE)

This section is only available when *Connect Type* is set to "0..Contact". The specified values for *Friction (FRIC)* and *Sep Force (FNTOL)* are written to the first "SLAVE" line of the BCTABLE entry, while the values for *Fric Limit Stress (FRLIM)* and *Hard-Soft Ratio (HARDS)* are written to the "FBSH" line.

Solid Contact, Shell Contact, and Contact Edges

These sections are used to specify how slave and master surfaces may come into contact. The options selected for the *Master* in each section will all be used together to write a particular value to the COPTM entry on the BCTABLE, while the options selected for the *Slave* in each section will all be used together to write a particular value to the COPTS field. For both, the value is determined in the following manner (example shows how COPTS is calculated for Slave regions, but the same applies to creating the value of COPTM for Master regions):

COPTS value = A+10*B+1000*C

where:

A = the value for Slave option specified in the Solids section (value is 0 or 1)

- 0 = *Slave* option is disabled, outside of solid elements in the Slave will not be considered for contact
- 1 = Slave option is enabled, outside of solid elements in the Slave will be considered for contact

B = the value for options specified in the Shell Contact section (value ranges from 0 to 6):

- 0 - *Slave* option is disabled, shell contact will not be considered for contact
- 1 - *Slave* option enabled, *Faces* set to "2..Top and Bottom", *Include Thickness* option enabled
- 2 - *Slave* option enabled, *Faces* set to "0..Bottom", *Include Thickness* option enabled
- 3 - *Slave* option enabled, *Faces* set to "0..Bottom", *Include Thickness* option disabled
- 4 - *Slave* option enabled, *Faces* set to "1..Top", *Include Thickness* option enabled
- 5 - *Slave* option enabled, *Faces* set to "1..Top", *Include Thickness* option disabled
- 6 - *Slave* option enabled, *Faces* set to "2..Top and Bottom", *Include Thickness* option disabled

C = the value for options specified in the Contact Edges Section (value is 0, 1, 10, or 11)

- 0 - *Slave* option is disabled, beam/bar or hard shell edges will not be considered for contact
- 1 - *Slave* option enabled, *Edges* set to "0..Beam Edges"
- 10 - *Slave* option enabled, *Edges* set to "1..Shell Edges"
- 11 - *Slave* option enabled, *Edges* set to "2..Beam and Shell Edges"

Advanced Options button

Opens the MSC Nastran Advanced Contact Options dialog box. Options in the *Glue Break* section are written to the appropriate fields on the "BKGL" line of the BCTABLE entry, while options in the *Contact* section are written to the appropriate fields on the "SEGS" line of the BCTABLE entry.

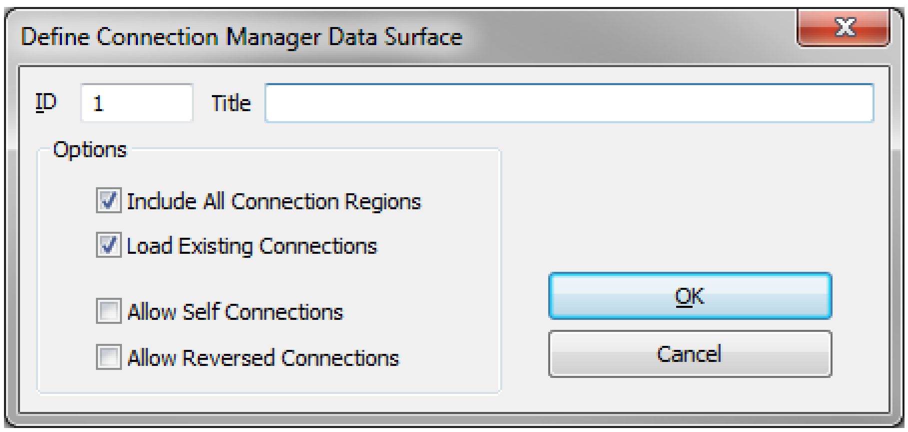
Refer to your MSC Nastran Quick Reference Guide for detailed descriptions of these options.

- Added *Penalty Autoscale* option to the *Common Contact (BCTPARM) and Glue (BGPARM) Parameters* section of the *NX Linear* tab of the *Define Connection Property* dialog box, which creates the AUTOSCAL field on the BCTPARM entry and scales the automatically calculated penalty factors PENN and PENT either up or down and can be used to scale the stiffness of specific contact pairs if convergence issues occur (Default = 1.0).

- Removed *Avg Method*, *Adaptive Stiffness*, *Penetration Factor*, and *Min Contact Percentage* options from the *Common Contact (BCTPARM) and Glue (BGPARM) Parameters* section of the *NX Linear* tab of the *Define Connection Property* dialog box, as these options are no longer documented in the NX Nastran Quick Reference Guide and should no longer be used.
- Added *Connection Manager Data Surface* to the *Data Surface Editor*, which allows you to manage connections in your model using an interactive “contact table”. All or any number of selected connection regions can be added to the “contact table”, while at the same time any existing connectors referenced by the connection regions can also be added. Once in the data surface, editing can take place.

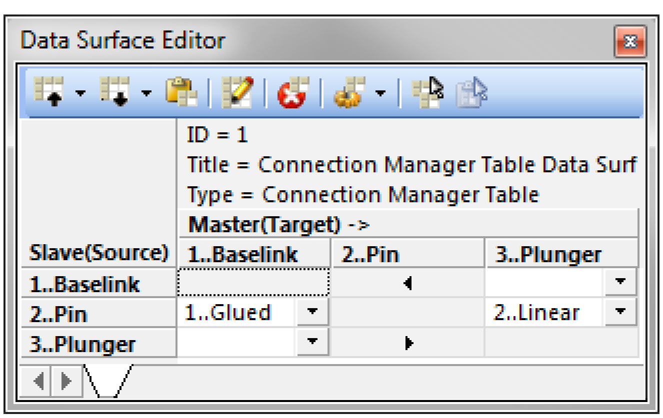
This Data Surface allows you to manage existing connections in a model or possibly create new connections using an interactive “contact table” format. For more information about connections (connection property, connection regions, and connectors), see Section 4.4, "Creating Connections and Regions".

The type of entities included in a *Connection Manager Data Surface* when it is created and the type of connections which are allowed in the table are all defined using the *Define Connection Manager Data Surface* dialog box:



When *Include All Connections Regions* is “on”, which is the default, all connection regions in the model will be added to the Data Surface. When the option is turned “off”, you need to select the connection regions of interest using the *Select Connection Regions in Data Surface* dialog box.

Each connector has a *Master (Target)* region and a *Slave (Source)* region which have a “connection” controlled by a specified connection property. When *Load Existing Connections* is “on”, which is the default, the connection property used by each connector will be shown as a drop-down control in the cell of the data surface corresponding to “*Master (Target)* region, *Slave (Source)* region”, while the cell corresponding to “*Slave (Source)* region, *Master (Target)* region” will contain an “arrow” symbol pointing in the direction of the *Master (Target)* region in the table. If either the Master (Target) region or Slave (Source) region of a connector has not been sent to the data surface, then no “*Master (Target)* region, *Slave (Source)* region” exists in the table, so it cannot be populated with a connection property. If *Load Existing Connections* is turned off, only empty drop-down controls containing the available connection properties will be shown in the contact table.



The *Allow Self Connections* option allows “self contact” to be defined in the contact table (i.e., a single connection region can be put into contact with itself to create a connector). Typically, this is not desired, but can be done with some solvers and this option allows you to select a connection property using a drop-down control at the “region #, identical region #” cell in the table, which is usually not available.

The *Allow Reversed Connections* option allows “reversed contact” to be defined in the contact table (i.e., in “connector A”, “region 1” is the *Master (Target)* region and “region 2” is the *Slave (Source)* region, while in “connector B”, “region 2” is the *Master (Target)* region and “region 1” is the *Slave (Source)* region). Typically, this is also not desired, but this option allows you to select a connection property for the "*Slave (Source)* region, *Master (Target)* region" cell instead of the cell showing an "arrow" symbol pointing towards the *Master (Target)* region.

After clicking OK in the *Define Connection Manager Data Surface* dialog box, the data surface will be populated based on the selected options. Using the defaults values for all options creates a data surface for a model with 3 connection regions and 2 connectors that will appear similar to this:

All connection regions sent to the data surface will be represented by both a column, which represents when the region is used as the *Master (Target)* in a connector, and a row, which represents when the region is used as the *Slave (Source)* in a connector. By default, the cell in the table which represents the same region as both *Master (Target)* and *Slave (Source)* will be "grayed", unless the *Allow Self Connections* option was used to initially create the data surface. The cells in the table which contain a connection property already selected in a drop-down control represent connections loaded from the model. The "arrow" icon in a cell can be used to swap the *Master (Target)* and *Slave (Source)* regions in a connection. Once all updates have been made, use *Create Connections* command.

Context Sensitive menus for Connection Manager Data Surface

Additional commands on the context-sensitive menu for column and row headers:

Show All Regions - makes all connection regions currently in the data surface visible.

Hide Selected Regions - temporarily removes both the row and column representing any connection region column or row currently highlighted in the data surface. Can also be used when cells are highlighted.

Auto Highlight (toggle) - when on, automatically highlights connection regions in the graphics window corresponding to a selected column or row as you select different columns or rows in the table. Also exists for cells.

Highlight - highlights connection regions in the graphics window corresponding to a selected column, selected row, or selected cells.

Additional commands on the context-sensitive menu for cells:

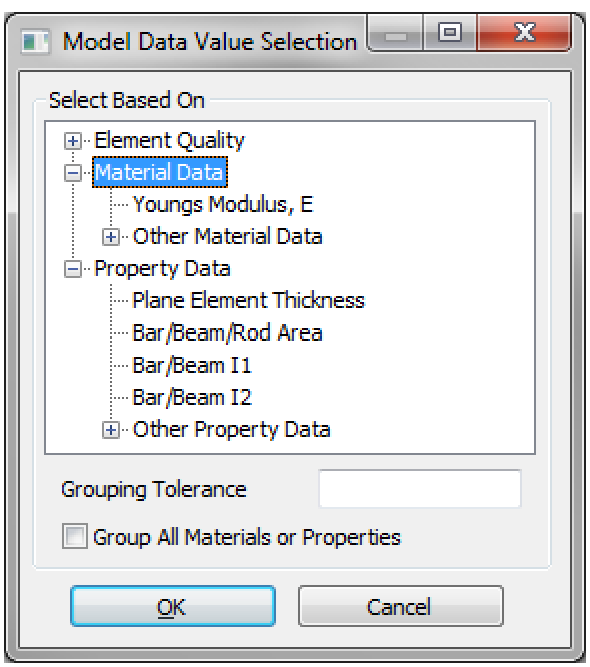
Show Connected Regions - makes all connection regions which are currently part of any connector visible in the data surface, while also hiding any connection region(s) which are not currently part of a connector.

Show Regions - allows you to select connection regions to make visible using a selection dialog box.

Revers - swaps the Master (Target) and Slave (Source) regions of the selected cell.

Create Connections - once all updates have been completed in the data surface, use this command to create any new connectors now defined in the data surface and also delete any connectors no longer defined.

Groups and Layers



•Added *Group, Operations, Generate Model Data Value* command, which automatically creates groups of elements using the value each element has for a selected type of model data, such as specific type of *Element Quality* check, a specific type of *Material Data*, or a specific type of *Property Data*.

Some common uses would be to create groups based on the different thicknesses of plane elements in a shell model, or the different *Youngs Modulus* values of the materials in an assembly model using solid elements, or the different values for the Aspect Ratio element quality check throughout the model. The type of model data is selected using the *Model Data Value Section* dialog box:

Simply choose a single *Element Quality* type or type of *Material Data/Property Data* by highlighting a specific value from the tree structure. All *Element Quality* values and most *Material Data/*

Property Data values in FEMAP should be available for selection, with the major exceptions being the material values of the materials of the “Other Types” material type.

There are 5 “special cases” which are a little more flexible with regard to the type of materials/properties in the model. For instance, “Plane Element Thickness” will work for any plane elements which have a thickness, including laminates. Others like “Bar/Beam/Rod Area” will allow you to create groups based on the areas on the properties referenced by all of the selected Beam, Bar, or Rod elements.

You can use the *Group Tolerance* to attempt to group entities with similar, but not identical, values together in a single group. Using the *Group All Materials or Properties* option simply considers all materials or all properties in the model, not only the materials and properties referenced by the selected elements.

- Added *Group, Solid, Type* command to add solids of the selected “solid type” (*Solid*, *Sheet*, *Wire*, or *General/NonManifold*) to the active group.

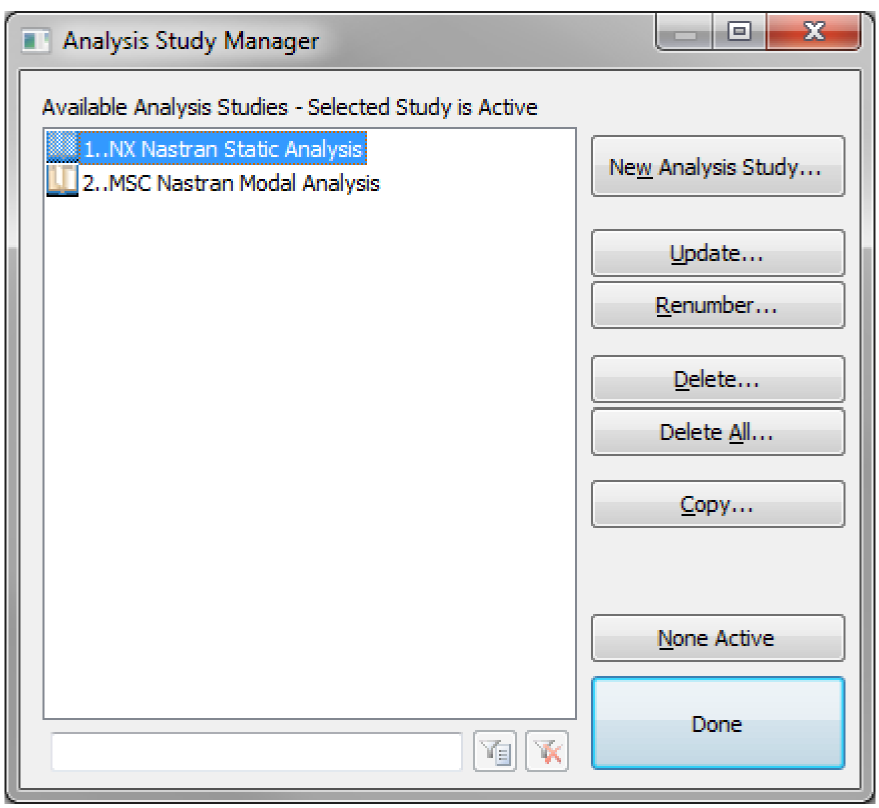
Views

- Updated the *View, Visibility* command by adding a *Hide...* button to the *Coord Sys*, *Geometry*, *Material*, *Property*, *Aero Panel/Body*, and *Aero Spline/Control Surface* tabs, which allows you to select entities in the graphics window to “hide” (i.e., turn off visibility) using the standard entity selection dialog box for that entity type.
- Updated “Element - Coordinate System” option in “Labels, Entities and Color” *Category* of *View, Options* command to with number of different element types. Also, added “2..RGB Line” to *Color Mode*.

Output and Post-Processing

- Added ability to show contour plots on line elements, so now it is possible to show results on line elements, planar elements, and solids elements in a single contour plot.
- Added *Model, Output, Create/Manage Analysis Study* command which creates a new analysis study or activates an existing analysis study.

An analysis study is a collection of output sets (individual sets of results data) which can be manually created at any time, then any number of existing output sets can be placed into the study at any time using various commands in the *Model Info Tree* dockable pane. See Section 8.5.2, "Model, Output, Create/Manage Set..." for more information about output sets.



Alternatively, studies may be automatically created when importing or attaching to results file(s) created by an analysis solver. This behavior is controlled using options on the *Results* tab of the *Preferences* dialog box, which is accessed by the *File, Preferences* command. For more information about automatic creation of studies and controlling the titles of those automatically created studies, see Section 2.6.2.8, "Results". Analysis Studies can also be automatically generated using *Add to New Study* in the *Model Info Tree* and/or using the *Result Set Processing Data Surface*.

To create a new analysis study, click the *New Analysis Study* button. In the *New Analysis Study* dialog box, you can use the default *ID* or specify your own *ID*, assign a *Title* to the study, select a *Solver* and *Analysis Type* for the study from the drop-down lists, and enter any additional information about the study in the *Notes* section.

The *Date* and time will also be automatically recorded when the study is created. The value for *Analysis Set* will be “Title” and the *Number Set* will be “0” for any study created using the “manual” method.

To activate an existing analysis study, highlight it in the list. Click *None Active* to deactivate all analysis studies.

Use *Update* to change the *Title*, *Solver*, *Analysis Type*, and/or *Notes* of the analysis study selected in the list, while *Renumber* will renumber the selected study. *Delete* will delete it from the model, but all of the output sets in the study will remain in the model. *Delete All* will delete all analysis studies from the model, but again, all of the output sets in the various studies will remain in the model. *Copy* will ask you to specify an ID, then create a copy of the selected analysis study. Only the information from the study itself, not the output sets in the study, will appear in the copy of the selected study.

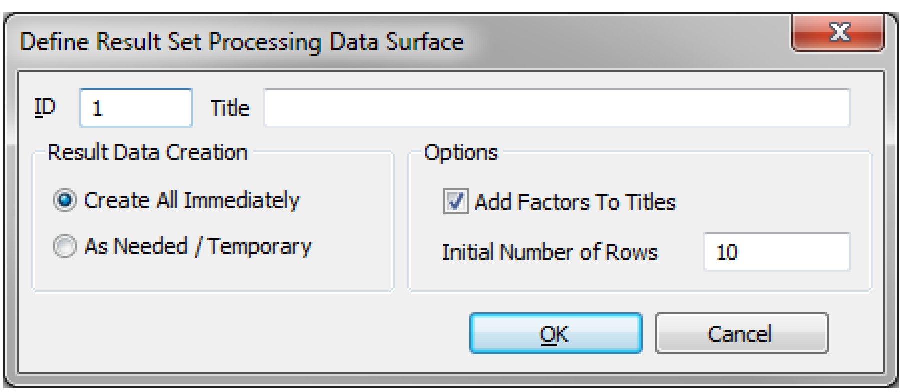
When a study is automatically created after a results file has been attached to or imported, the *ID*, *Title*, *Solver*, *Analysis Type*, *Analysis Set* (if analysis model was created in FEMAP), *Date*, and *Number Sets* will be populated using information attained from the results file at the time of attachment or import.

- Added *Result Set Processing Data Surfaces* to the *Data Surface Editor*, which allows you to create to automatically create any number of new output sets based on existing output sets in the model. The scale factors for each existing load set may be entered into the *Data Surface Editor* directly or pasted in from another program, such as excel.

This Data Surface has some very specific uses, which involve creating new output sets using existing output sets or updating output sets previously created using this type of Data Surface. These output sets are created or updated using the “Linear Combination”, “RSS Combination”, or “Envelope” methods also available in the *Model, Output, Process* command. See Section 8.5.6, "Model, Output, Process" for more information on these methods.

Any combination of the “Linear Combination”, “RSS Combination”, and/or the various “Envelope” methods may be used in a single Data Surface to create or update previously created output sets.

Settings for this type of Data Surface are specified using the *Define Result Set Processing Data Surface* dialog box:



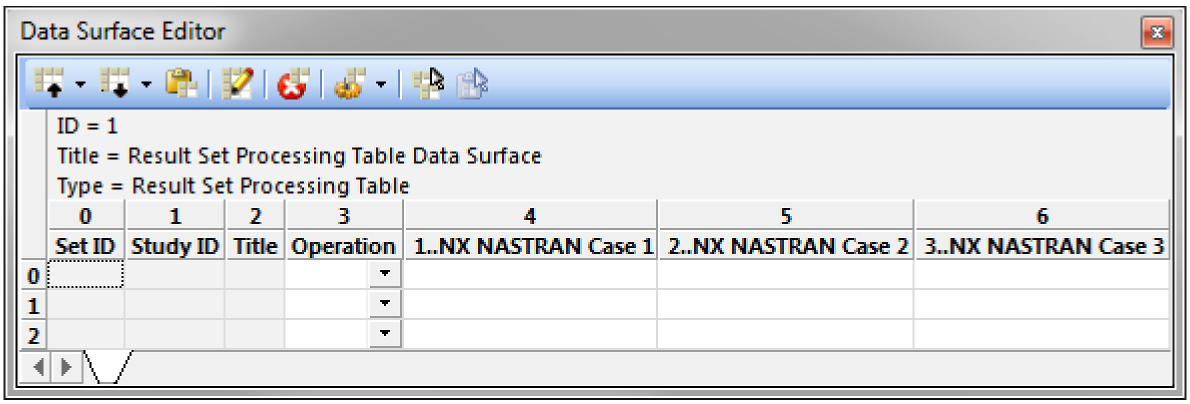
Use the *Results Data Creation* option to choose the method for creating the output set(s), *Create All Immediately* or *As Needed/Temporary*. When using *Create All Immediately* the output values will be stored in the created or updated output set(s). Output sets created with this option will be displayed with a “green closed envelope” icon. When using *As Needed/Temporary* the information about the combination(s) or envelope(s) is stored with the new or updated output set(s), but the actual output values are calculated “on-the-fly” from the individual output sets referenced by the new or update output set(s). Output sets created with this option will be displayed with a “yellow open envelope” icon.

In the *Options* section, turn the *Add Factors To Titles* option "on" to append the title of each new or updated output set with ('scale factor'* 'existing output set ID', etc.) for each existing output set used by the new or updated output set. If nothing is specified for *Title* in a particular row and this option is 'on', then the title will be "'Type of Operation' ('scale factor'* 'existing output set ID', etc.)". If no *Title* is given and this option is 'off', then the Output Set title will simply be 'Type of Operation'.

The *Initial Number of Rows* option is used to specify the number of rows, representing output sets to create, which will initially appear in the *Data Surface Editor* after the existing output sets have been selected.

Note: Only rows which have an Operation selected and contain a scale factor in at least one column will be used to create new output sets. If a row is blank, it will simply be skipped.

After clicking OK in the *Define Result Set Combination Data Surface* dialog box, select the existing output sets to possibly use in combinations using the *Select Output Sets to Use in Processing* dialog box, then click OK. The *Data Surface Editor* will now appear like this:



A *Set ID*, *Study ID*, and *Title* may optionally be entered for each row. To include an existing output set in a new output set, simply enter a scale factor on a specific row in the appropriate column. Use 1.0 to simply include the existing output set with no scaling. Any row which has scale factors defined must also have an *Operation* selected, otherwise an error will be issued.

Available options for *Operation* correlate to different methods available when using the *Model, Output, Process* command and include *Linear Combination*, *RSS Combination*, *Max Envelope*, *Min Envelope*, and *MaxAbs Envelope* (Maximum Absolute Value). The *SetID Max Envelope*, *SetID Min Envelope* and *SetID MaxAbs Envelope* options are typically used in conjunction with the "envelope" *Operation* of the same type and create output sets containing only output set IDs, similar to using the *Store Set/Location Info* option in *Model, Output, Process*.

If you specify a *Study ID* for a particular row, then the output set created or updated by that row will be added to the specified Analysis Study. Once all of the scale factors have been entered, click the right-mouse button in any cell and choose the *Create Processed Results Sets* command from the context-sensitive menu to create new output sets or update existing output sets specified in the *Set ID* column.

Note: If no values are entered for *Set ID*, the next available load set ID(s) will be used for the new output sets when they are created. If any values for *Set ID* are the same as existing output set IDs in the model, a question will be asked, "Ok to Delete Existing Result Sets? Data Surface references Result Sets that already exist. Press Yes to Delete and Recreate them, No to Create New Sets." If any values for *Set ID* are the same as an output set currently represented by a column in the *Data Surface Editor*, then a question will be asked, "Ok to Combine? Combining will delete Result Sets used in this Data Surface and you may get no Output." Typically, this question should be answered "No".

Note: If no values are entered for *Study ID* and all of the output sets to be process exist in the same Analysis Study, then any new or updated output sets will also be added to that Analysis Study. If no values are entered for *Study ID* and the output sets to be processed exist in different Analysis Studies, then a value of 0 is used for *Study ID* and the new or updated output sets will not be added to any Analysis Study.

Context Sensitive menus for Result Set Processing Data Surface

Additional commands on the context-sensitive menu for cells:

Fill All Sets - applies a single user-defined scale factor value to all output sets in all highlighted rows.

Apply to Study - if a data surface contains only output sets from a single Analysis Study, then the *Title*, *Operation*, and scale factors specified for the output sets can potentially be "applied" to a different Analysis Study. This command will only be successful when the selected Analysis Study contains the same number of output sets that are currently referenced by the data surface. When successful, the output set columns currently in the *Data Surface Editor* are replaced with the corresponding output sets from the selected Analysis Study. You can then use *Create Processed Results Sets* to create new output sets in the selected Analysis Study.

Create Processed Result Sets - used to create the new output sets. All output set processing currently defined in the *Data Surface Editor* will be done at the same time.

Additional commands on the context-sensitive menu for column headers:

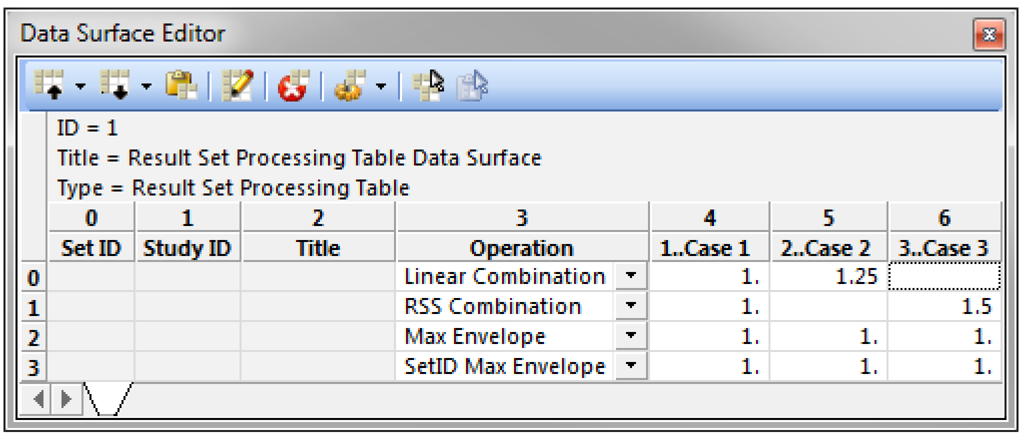
Add Result Sets - allows you to add columns by selecting any number of output sets using a dialog box

Change Result Set - allows you to update the output set referenced by a particular column

Remove Selected Result Sets - allows you to remove an output set currently represented by a column

Example

This type of Data Surface can be used to create 4 new output sets, each using a different Operation (a *Linear Combination*, a *RSS Combination*, a *Max Envelope*, and a *SetID Max Envelope*), using 3 existing output sets. Simply enter scale factor values into various cells. It is typical to simply use 1.0 for envelope operations, but you can enter other scale factors and those scaled values will then be used to determine the envelope values:

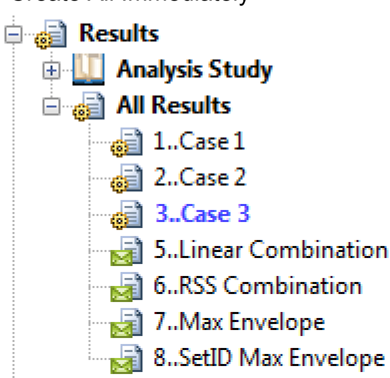

Data Surface Editor

ID = 1
Title = Result Set Processing Table Data Surface
Type = Result Set Processing Table

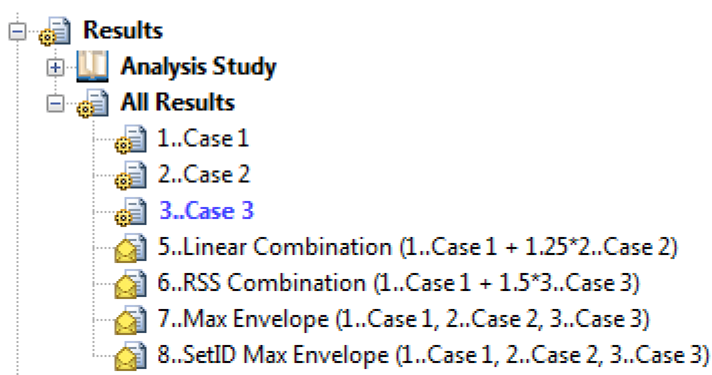
	0	1	2	3	4	5	6
	Set ID	Study ID	Title	Operation	1..Case 1	2..Case 2	3..Case 3
0				Linear Combination	1.	1.25	
1				RSS Combination	1.		1.5
2				Max Envelope	1.	1.	1.
3				SetID Max Envelope	1.	1.	1.

The *Set ID* and *Title* fields are not defined, therefore, the next available Output Set IDs will be used along with "Default" Titles.

Titles Undefined
Add Factors To Titles = Off
Result Data Creation =
Create All Immediately



Titles Undefined
Add Factors To Titles = On
Result Data Creation = As Needed/Temporary



If all referenced output sets are in the same Analysis Study, then any new output sets created or updated will also be placed into that Analysis Study, unless a *Study ID* is defined in the *Data Surface Editor*. If the referenced output sets are from different Analysis Studies, then they will only be placed into an Analysis Study if a *Study ID* is defined. If an Analysis Study is referenced in *Study ID* but does not exist, a new Analysis Study will be created.

- Added *Section Cut* as a *Display Mode* option for the *Freebody Tool* in the *PostProcessing Toolbox*. A user-defined “cut plane” is used to automatically determine the *Freebody Nodes* and *Freebody Elements* to essentially create an *Interface Load*. The “cut plane” and “path” can be defined using one of four methods and the location of the *Total Summation Vector* typically moves along a specified “path”. A number of additional options are available to control the behavior of the *Section Cut*, including the ability to select elements on the reverse side of the plane, control inclusion of elements “cut” by the “cut plane”, etc.

Entities (Section Cut) - When *Display Mode* is set to *Section Cut*, the *Entities* section has quite a number of different options compared to when *Display Mode* is set to *Freebody* or *Interface Load*:

Display Mode = Section Cut (Entity Selection Mode)

⊟ Entities	
⊟ Entity Selection Mode	Plane / Vector
Cutting Plane Tolerance	0.
Reverse Cutting Plane	☐
Search Radius	0.
Limit Entities to Group	☐
Include Clipped	☐
⊞ Plane	[0.,0.,0.] [0.,0.,1.]
⊞ Vector	[0.,0.,0.] [1.,0.,0.]

Entity Selection Mode (Section Cut) - allows you to select one of four options from a drop-down list to control how the plane and/or “path” for the plane to follow will be specified.

Plane/Normal (default) - allows you specify a plane using the standard plane definition dialog box, then the plane simply follows a vector normal to the center of the specified plane.

Plane/Vector - allows you specify a plane using the standard plane definition dialog box and also specify a vector using the standard vector definition dialog box to use as the “path” at the center of the plane.

Vector - allows you to specify a vector as the “path” for the plane to follow with the plane being automatically determined using the specified vector as the normal vector at the center of the plane.

Curve - allows you to select a curve to use as the “path” for the plane to follow with the plane being automatically determined using the normal vector calculated at the current location along the selected curve.

Note: Depending on the selected option, a different field or fields will appear below the *Include Clipped* option which will allow you to change the specified *Plane*, specified *Vector*, or selected *Curve ID*.

Cutting Plane Tolerance (Section Cut) - allows you to enter a “tolerance” value which essentially gives the *Section Cut* plane “thickness”. This “thickness” extends from the specified plane in the positive direction only. Clicking the “Calculate tolerance based on model size” icon button will return a value of 1/1000 of the overall model box diagonal. Default value is 0.0.

Note: The *Cutting Plane Tolerance* is designed to be used in cases where nodes are very close to planar, but actually do not all lie on the same plane. To include more nodes and elements on the positive side of the specified *Section Cut* plane, you should move the plane to a more appropriate location.

Reverse Cutting Plane (Section Cut) - when enabled, reverses the direction of the specified *Section Cut* plane.

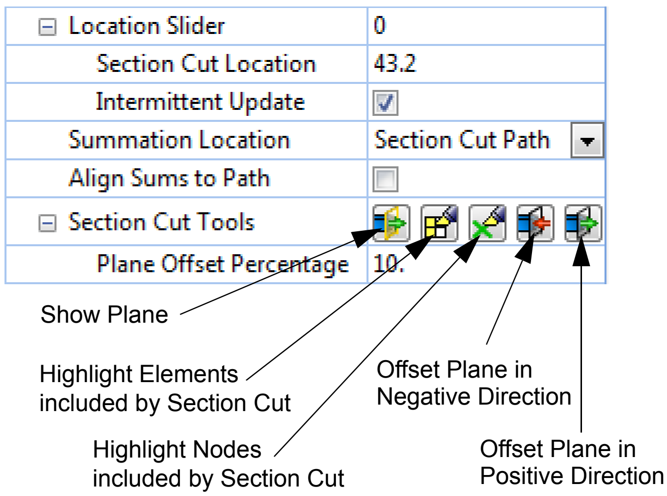
Search Radius (Section Cut) - allows you to only consider nodes and elements within the specified search radius from the center of the *Section Cut* plane. Use the *Measure* icon button to use the measured distance between two locations as the search radius.

Limit Entities to Group (Section Cut) - when enabled, allows you to only consider nodes and elements found in the active group or a selected group.

Note: It is recommended to have both nodes and elements in the group, but if no elements are in the group, then all elements will be considered *Peripheral Elements*.

Include Clipped (Section Cut) - when enabled, any elements "cut" by the *Section Cut* plane will be included in the calculation of the *Interface Load*. When disabled, any "cut" elements will be ignored.

Display Mode = Section Cut (Other Entities Options)



Location Slider (Section Cut) - slider control used to change the position of the *Section Cut* plane from 0% to 100% along the specified "path". You can also enter a value directly to the left of the slider control. When the *Intermittent Update* option is enabled, the values for *Total Summation Vector* and *Nodal Vector(s)* will dynamically update as the slider control is moved to the left or right. When *Intermittent Update* is disabled, the values for *Total Summation Vector* and *Nodal Vector(s)* will only be calculated once you stop moving and then let go of the slider control

Note: The *Section Cut Location* value is determined using a combination of the original location of the specified plane and/or how the "path" is specified. It changes as the slider control is moved left or right, but is not really recommended for the user to enter this value directly.

Summation Location (Section Cut) - allows you to select one of three options from a drop-down list to control to location of the *Total Summation Vector*.

Section Cut Path (default) - location of the *Total Summation Vector* moves along the specified "path".

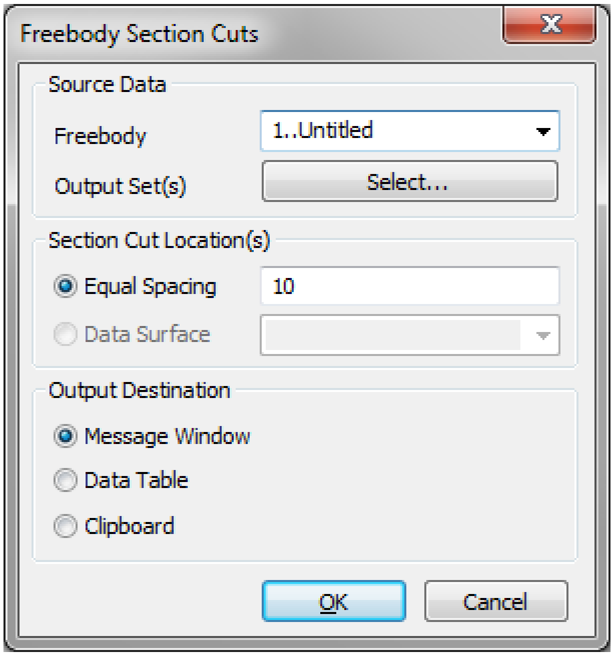
Freebody Node Centroid - location of the *Total Summation Vector* will be positioned at the centroid of the nodes automatically selected by the *Section Cut* plane.

Static Location - location of the *Total Summation Vector* stays in a single static location and needs to be specified using the *Location* field in the *Total Summation Vector* section.

Align Sums to Path (Section Cut) - aligns the X-component of the *Total Summation Vector* and *Nodal Vector(s)* to the normal vector of the plane for *Entity Selection Mode* set to *Plane/Normal*, to the specified vector for *Entity Selection Mode* set to *Plane/Vector* or *Vector*, or a vector tangent to the curve at the specified location along the curve when *Entity Selection Mode* set to *Curve*. The Y and Z-components are then determined via cross products.

Section Cut Tools (Section Cut) - section contains 5 icon buttons, each with a specific purpose. From left to right, the icon buttons *Show Plane* in the graphics window, *Show Freebody Elements* or *Show Freebody Nodes* currently included in the *Section Cut* by highlighting them in the graphics window, *Offset Section Cut Plane in Negative Direction* or *Offset Section Cut Plane in Positive Direction* using the *Plane Offset Percentage* value.

- Added List, Output, Freebody Section Cut command, which is only available when at least one Freebody entity with *Display Mode* set to *Section Cut* exists in the model (See Section 7.2.3.3, "Freebody tool"). Creates a listing and sends it to the specified *Output Destination*, using the selected Freebody entity and additional options specified by the user in the *Freebody Section Cuts* dialog box:



Source Data - used to select which freebody entity and output sets will be used to create the listing

Freebody - Select an existing Freebody entity using this drop-down. Only Freebody entities with *Display Mode* set to *Section Cut* can be selected.

Output Set(s) - Press the *Select* button to choose any number of output sets using the standard *Select Output Set(s)* dialog box. For more information, see Section 8.6.0.1, "Using the Select Output Sets and Select Results dialog boxes". After output set selection is complete, the text on Select button will change to be "# Output Set(s) Selected".

Section Cut Locations - offers two different methods for specifying cut locations. An individual row with values corresponding to each cut location will be listed to the *Output Destination*. The values at all cut locations will be listed together for each output set. For example, if 10 cut locations are specified and 2 output sets are selected, the first 10 rows will be the values at the 10 cut locations using output set 1, then the next 10 rows will be the values at the 10 cut locations using Output Set 2.

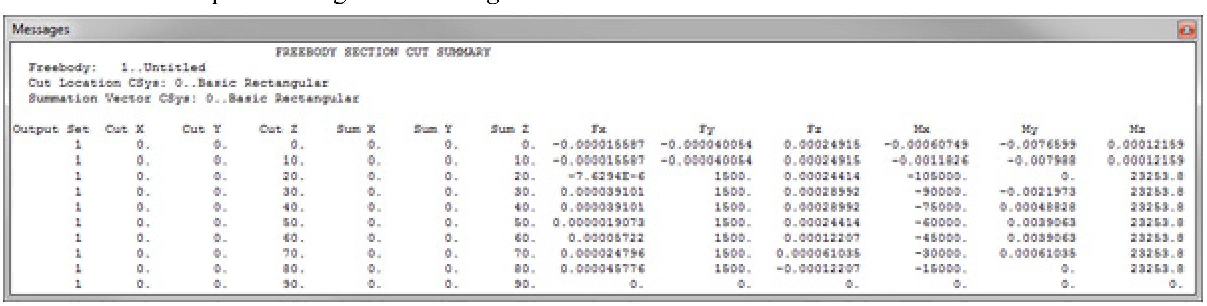
Equal Spacing - When using this option, the value represents the number of equally spaced instances of the specified plane to be used as cut locations.

Data Surface - When using this option, use the drop-down to select an existing Along Coordinates Data Surface. Only the XYZ *Location* values from the Along Coordinates Data Surface will be used. If the Freebody entity has Entity Selection set to Curve, then only the values in the X *Location* column will be used and must be between 0.0 (0% or curve length) and 1.0 (100% of curve length).

Output Destination - used to select where the listing will occur, the *Message Window* (*Messages* dockable pane), the *Data Table* (dockable pane), or the *Clipboard*.

When *Output Destination* is set to *Message Window*, the listing will use the following format for each cut location: *Output Set*, *Cut X*, *Cut Y*, *Cut Z*, *Sum X*, *Sum Y*, *Sum Z*, *Fx*, *Fy*, *Fz*, *Mx*, *My*, and *Mz*.

Here is an example of listing to the *Message Window*:


Messages

FREEBODY SECTION CUT SUMMARY

Freebody: 1..Untitled
Cut Location CSys: 0..Basic Rectangular
Summation Vector CSys: 0..Basic Rectangular

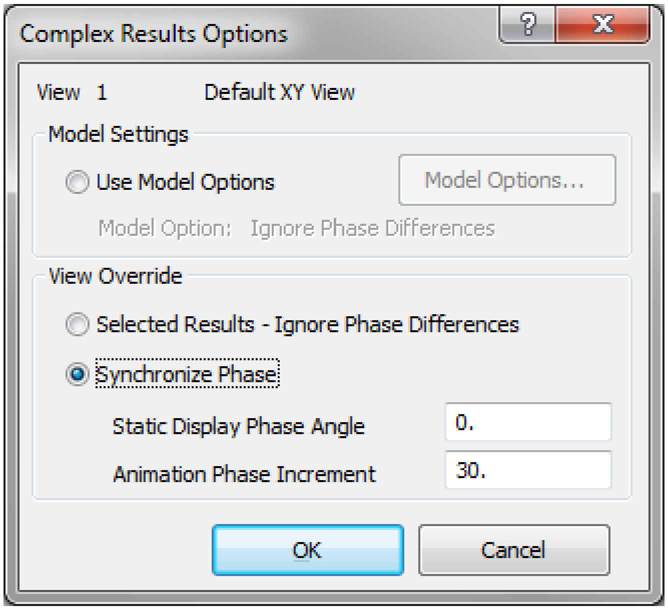
Output Set	Cut X	Cut Y	Cut Z	Sum X	Sum Y	Sum Z	Fx	Fy	Fz	Mx	My	Mz
1	0.	0.	0.	0.	0.	0.	-0.000015687	-0.000040054	0.00024915	-0.00060749	-0.0076599	0.00012159
1	0.	0.	10.	0.	0.	10.	-0.000015687	-0.000040054	0.00024915	-0.0011826	-0.007988	0.00012159
1	0.	0.	20.	0.	0.	20.	-7.6294E-6	1500.	0.00024414	-105000.	0.	23253.8
1	0.	0.	30.	0.	0.	30.	0.000039101	1500.	0.00028992	-90000.	-0.0021973	23253.8
1	0.	0.	40.	0.	0.	40.	0.000039101	1500.	0.00028992	-75000.	0.00048828	23253.8
1	0.	0.	50.	0.	0.	50.	0.0000019073	1500.	0.00024414	-60000.	0.0039063	23253.8
1	0.	0.	60.	0.	0.	60.	0.00008722	1500.	0.00012207	-45000.	0.0039063	23253.8
1	0.	0.	70.	0.	0.	70.	0.000024796	1500.	0.000061035	-30000.	0.00061035	23253.8
1	0.	0.	80.	0.	0.	80.	0.000045776	1500.	-0.00012207	-15000.	0.	23253.8
1	0.	0.	90.	0.	0.	90.	0.	0.	0.	0.	0.	0.

When *Output Destination* is set to *Data Table* or *Clipboard*, the listing will use the following format for each cut location: *Set ID* (Output Set ID), *Cut CS* ("Cut Location" Coordinate System), *Cut X*, *Cut Y*, *Cut Z*, *Sum X*, *Sum Y*, *Sum Z*, Output CS ("Summation Vector" Coordinate System), *Fx*, *Fy*, *Fz*, *Mx*, *My*, and *Mz*.

Note: If using *Output Destination* set to *Clipboard*, it should be noted that the values will be pasted into individual cells in a spreadsheet program like Microsoft Excel.

- Updated the *View, Select* command by adding the *Complex Results* button to the *Select PostProcessing Data* dialog box, which is accessed by clicking the *Deformed and Contour Data* button. These options enable you to set overall "Model Option(s)" or view-specific "override" values used to convert complex data to real data, on-the-fly, when post-processing complex results.

In the *Model Settings* section, when *Use Model Options* is selected, the current setting for “Model Option” is displayed. To change the “Model Option”, click the *Model Options* button to display a dialog box with two options:



Selected Results - Ignore Phase Differences (default) - displays selected results using only the magnitude (i.e., ignores phase component).

Synchronize Phase - displays selected results using a specified *Static Display Phase Angle* and/or *Animation Phase Increment.*

Static Display Phase Angle - specified phase angle value used for calculation of results when displaying a deformed or contour plot. When used, the specified phase angle value will be added to the end of the Post Title for reference.

Animation Phase Increment - specified phase increment used to effectively creating a range of phase angles, which are then used to create an appropriate number of animation frames to animate complex results. When used, the phase angle value used to create each animation frame will be shown at the end of the post title for reference.

These same options are available in the *View Override* section, only the values specified in this section will only be used for the current view instead of the entire model.

Note: This functionality is similar to the functionality of the *Model, Output, Expand Complex* command, but does not need to create additional output sets, as all values required for displaying the results are calculated on-the-fly and then displayed. See Section 8.5.13, “Model, Output, Expand Complex...” for more information about working with complex results.

- Added “3..Material Direction” as an option for *Output Orientation* when transforming output. This can now be set using the drop-down control for *Forces*, *Stresses,* and *Strains* in *Tria3*, *Tria6*, *Quad4*, and *Quad8* sections.
- Added *Max Threshold* and Max/Min Threshold options to Level Mode for the “Contour/Criteria Levels” option in “PostProcessing” *Category* of *View, Options* command. *Max Threshold* will use the value specified for *Maximum* as a “threshold” value, thus all results above this value are shown using the color of the uppermost band of the *Contour/Criteria Legend*, while *Max/Min Threshold* does essentially the same thing, but uses both the *Maximum* and *Minimum* values as upper and lower threshold values.

Note: Both *Max Threshold* and *Max/Min Threshold* require the “Contour Fill Mode” to be set to “Level Colors”. Using *View Options*, this can be set using the *Contour Type* option (*Contour Fill Mode* set to “1..Level Colors”) or via the *Contour Options* dialog box, which can be accessed from various places, including the *Select PostProcessing Data* dialog box, the *Post Options* icon menu of the *Post* toolbar, or the *Contour Type* or *Contour/Criteria Style* options in *View Options*. It can be also be set via the *PostProcessing Toolbox* (*Contour* Tool - *Options* - *Levels* - *Continuous Colors* option disabled).

Geometry Interfaces

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

FEMAP Interface	Latest Supported Version
Parasolid	Parasolid 27.1
Solid Edge	Solid Edge with Synchronous Technology 7
NX	NX 10.0
CATIA	CATIA V5-6R2014
ACIS	ACIS 24, SP1
SolidWorks	SolidWorks 2015

- Updated *File, Import, Geometry* command by adding the *Sew Sheets into Manifold Solids* option to the *Solid Model Read Options* dialog box when import ACIS geometry (*.sat files). This option controls if sheets (surfaces) should be stitched into manifold solids. If this option is on, then the surfaces will only be "sewn" together into manifold solids. It will only create "manifold solids", as ACIS does not support "NonManifold Solids" (i.e., General Bodies) like Parasolid. If off, no "sewing" will occur.

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

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

FEMAP Neutral File Interface

- Updated Neutral Read and Write for v11.2 changes

NX Nastran Interface

- Added *GPU Computing* option to *NASTRAN and Solutions Options* dialog box. When enabled, writes GPGPU=ANY to the command line which instructs NX Nastran to automatically determine if a device with GPUs exists and, if so, to use it for during the solve.
- Added *SWPANGLE* option to *NASTRAN Bulk Data Options* dialog box. When enabled, allows you to enter the angular increment in degrees at which failure indices and strength ratios are computed and output for laminates in direct frequency (SOL 108) and modal frequency (SOL 111) analysis.
- Added *MGRID* and *MDOF* option to *NASTRAN Bulk Data Options* dialog box. When enabled, used to specify a specific node (MGRID) and degree of freedom (MDOF = 1, 2, 3, 4, 5, or 6) to monitor during a direct frequency or direct transient response, plotted in the *NX Nastran Analysis Monitor.*
- Added *Use NXN v8.5 Elastic Beam Formulation* option to *NXSTRAT Solver Parameters* dialog box. When enabled, (BEAMALG = 1), the algorithm for elastic beam formulation from NX Nastran 8.5 is used instead of the current algorithm for elastic beam formulation.
- Added *PYR_DETJ* option to *GEOMCHECK* dialog box, which is the Jacobian Determinant for pyramids. Also, added the *Include Parabolic Plate Checks* option, which when enabled, includes parabolic element checks using the same options and values specified for the linear versions of QUAD SKEW (Q8_SKEW), QUAD IAMIN (Q8_IAMIN), QUAD IAMAX (Q8_IAMAX) and TRIA IAMAX (TA6_IAMX).

- Added support for reading ply-by-ply stress and strain output from Direct Transient (SOL 109), Direct Frequency (SOL 108), Modal Transient (SOL 112), or Model Frequency (SOL 111) analysis for laminate elements created. This output was not available until NX Nastran 10.0.
- Added support for reading GPF output for contact from Advanced Nonlinear (SOL 601), which will appear as the Total Contact Force and Total Contact Moment vectors, plus their components.

A number of bugs were corrected

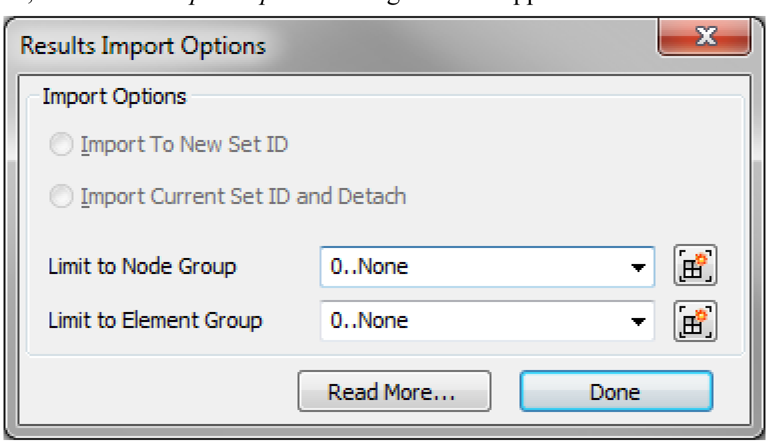
For details, see "Analysis Program Interfaces" in the *FEMAP User Guide*.

Nastran Interfaces (NX and MSC/MD)

- Added support for reading and writing of the ACCEL1 entry. Also, ACCEL1 entries can be created by applying an acceleration load to a node or geometry, then exporting a Static analysis.
- Added support for reading and writing of element corner thickness (TFLAG, T1, T2, T3, and T4 fields) for the CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR, CPLSTS3, CPLSTS4, CPLSTS6, and CPLSTS8 elements. Element thickness can be updated via the *Modify, Update Elements, Adjust Plate Thickness/Offset* and *Modify, Update Elements, Midsurface Thickness and Offset* commands.
- Added support for reading both Linear and Nonlinear results from a nonlinear analysis into the same output set. Previously, you would have to choose one or the other, but now both are read in during a single import or attach.
- Added *SECOMB* option to *NASTRAN Bulk Data Options* dialog box. When enabled, used to control if output will be combined for a superelement analysis, has a number of caveats.
- Added *ALPHA1* and *ALPHA2* options to *NASTRAN Bulk Data Options* dialog box. When enabled, *ALPHA1* is the complex scale factor applied to the mass matrix and *ALPHA2* to the stiffness matrix. Used in frequency and transient response analysis, if PARAM,ALPHA1 and/or ALPHA2 are not equal to complex zero, then Rayleigh's damping is added to the viscous damping.
- Added Label field in the *Master Requests and Conditions* and *Analysis Case* dialog boxes which can be used to write the LABEL entry in Case Control.
- Updated processing of NASTRAN Include to allow the INCLUDE statement to start in any column. Previously it had to start in column 1.
- Updated *File, Import, Analysis Results* when importing a XDB file.

After you run Nastran, choose the *File, Import, Analysis Results* command, select *Nastran*, and then choose *NX Nastran* or *MSC/MD Nastran* from the drop-down list. FEMAP will display the standard file access dialog box for you to choose the XDB file you want to read. When you press *OK*, FEMAP will immediately open the *Select Output to Internalize* dialog box, which facilitates selection of output sets and vectors. By default, all output sets in the *Output Sets* section will be selected and the *All Output Vectors* option in the *Output Vectors* section will be enabled, thus all output from the XDB file will be imported. To only import a subset of the results from the XDB file, simply select the desired output set(s) and optionally disable the *All Output Vectors* option to be able to select individual output vector(s) for import. For more information on using the *Select Output to Internalize* dialog box see Section 8.6.0.1, "Using the Select Output Sets and Select Results dialog boxes" in the *Commands* manual.

Once you click OK, the *Results Import Options* dialog box will appear:



In addition to selecting which output sets and output vectors to internalize, groups limiting output to certain nodes and/or elements may be selected. The “quick group” icons next to the drop-down lists can be used to create a new group or edit an existing group “on-the-fly”. Depending on which button you select, you will be able to choose only nodes or elements. If you click the *Read More...* button, the *Select Output to Internalize* dialog box will be displayed again for selection of additional output set(s) and output vector(s) and this process can be repeated as many times as needed.

A number of bugs were corrected

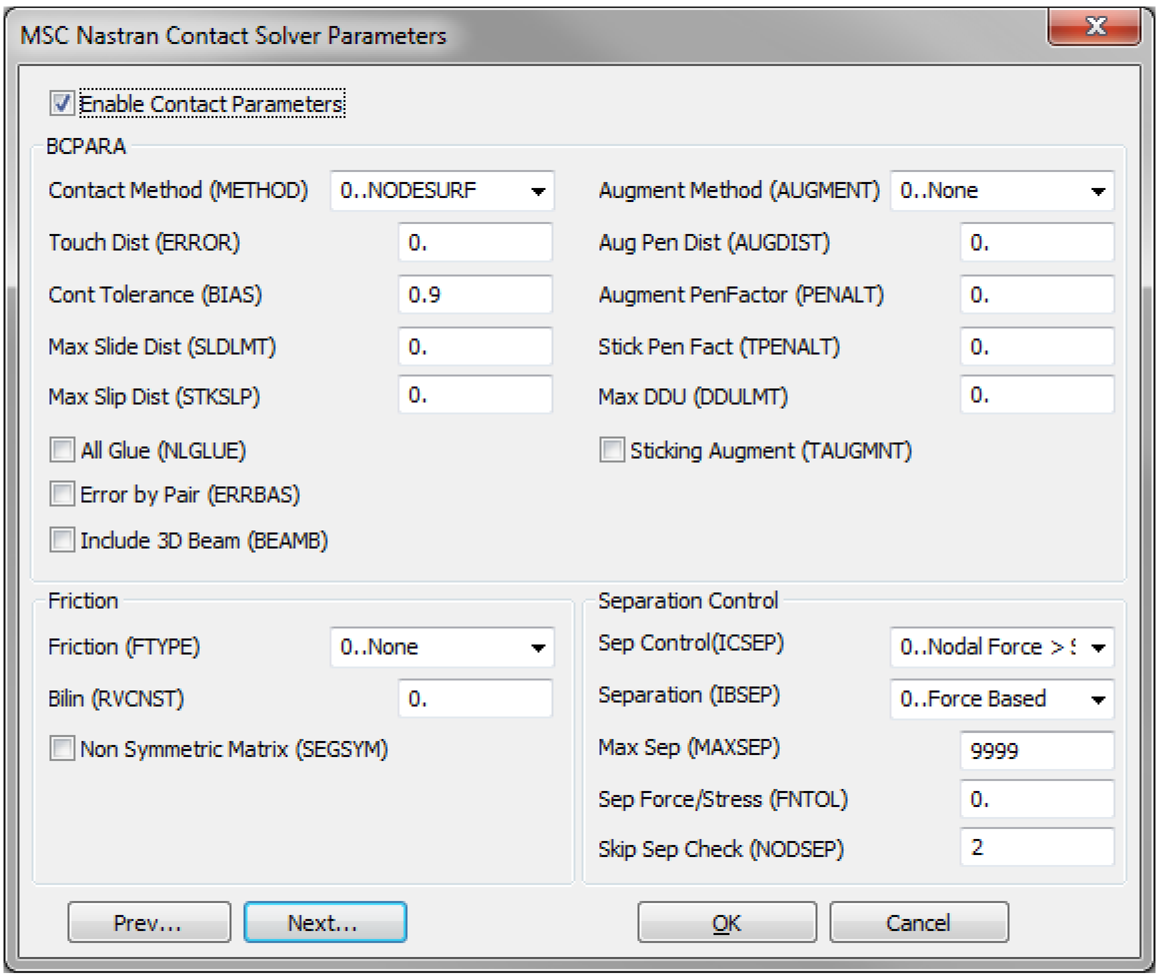
For details, see “Analysis Program Interfaces” in the *FEMAP User Guide*.

MSC/MD Nastran Interface

- Added *MSC Nastran Contact Solver Parameters* dialog box to Analysis Set Manager for Static analysis.

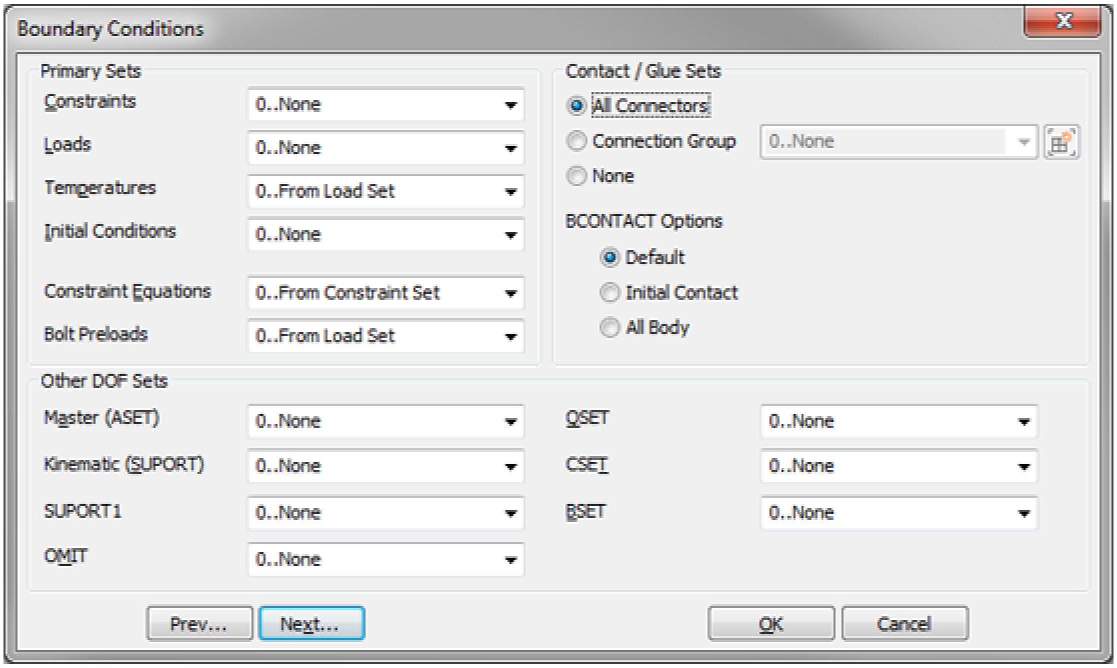
The *MSC Nastran Contact Solver Parameters* dialog box contains a number of options which can be specified when performing an analysis in MSC Nastran, which also includes surface-to-surface contact. All of the items in this dialog box correspond to items (Params) which can be included on the BCPARA entry. For more information, see the BCPARA (Contact Parameters - SOLs 101 and 400) entry in the MSC Nastran Quick Reference Guide.

You must check the “Enable Contact Parameters” box for FEMAP to use any options in this dialog box:



- Added *Nonlinear Options* dialog box to *Analysis Set Manager* for Static analysis. This dialog box is available in the *Master Requests and Conditions* section and individual subcases for static analysis when *Analysis Program* is set to MSC Nastran, but should only be used if the model contains linear contact. In that case, the *Enable NLPARM* option must be turned on for the subset of available options to be written to the input file.

- Added *Contact/Glue Sets* section to Boundary Conditions dialog box:



The upper portion of this section allows you to select which connectors in the model will be written to the BCTABLE entry in the MSC Nastran input file and if they will be used:

All Connectors (default) - simply writes all connectors currently enabled in the model to the input file.

Connection Group - allows you to select a group containing any number of connectors from the drop-down control, then writes out only those connectors. The *Quick Group* icon button can be used to quickly create a new group, then you can choose *Edit Group* to select connectors in the model to add to the selected group.

None - writes all enabled connectors out to the input file, but does not write BCONTACT to Case Control.

The *BCONTACT Options* portion controls how the BCONTACT entry is written in Case Control:

Default - for *All Connectors*, writes BCONTACT=107 and sets ID of BCTABLE entry t to 107, while for *Connection Group*, writes BCONTACT=108 and sets ID of BCTABLE entry t to 108. Can be used for either Linear or Glued contact.

Initial Contact - writes BCONTACT=0 and sets ID of the BCTABLE entry to 0. Only for Linear contact.

All Body - writes BCONTACT=ALLBODY and the BCTABLE is not written at all. Only for Linear contact.

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

ANSYS Interface

- Added reading and writing of linear and parabolic pyramid elements.
- Added reading Plastic Strain results from nonlinear analysis.
- Updated writing of coordinate systems and nodes to provide a higher level of precision.

A number of bugs were corrected.

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

ABAQUS Interface

A number of bugs were corrected.

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

DYNA Interface

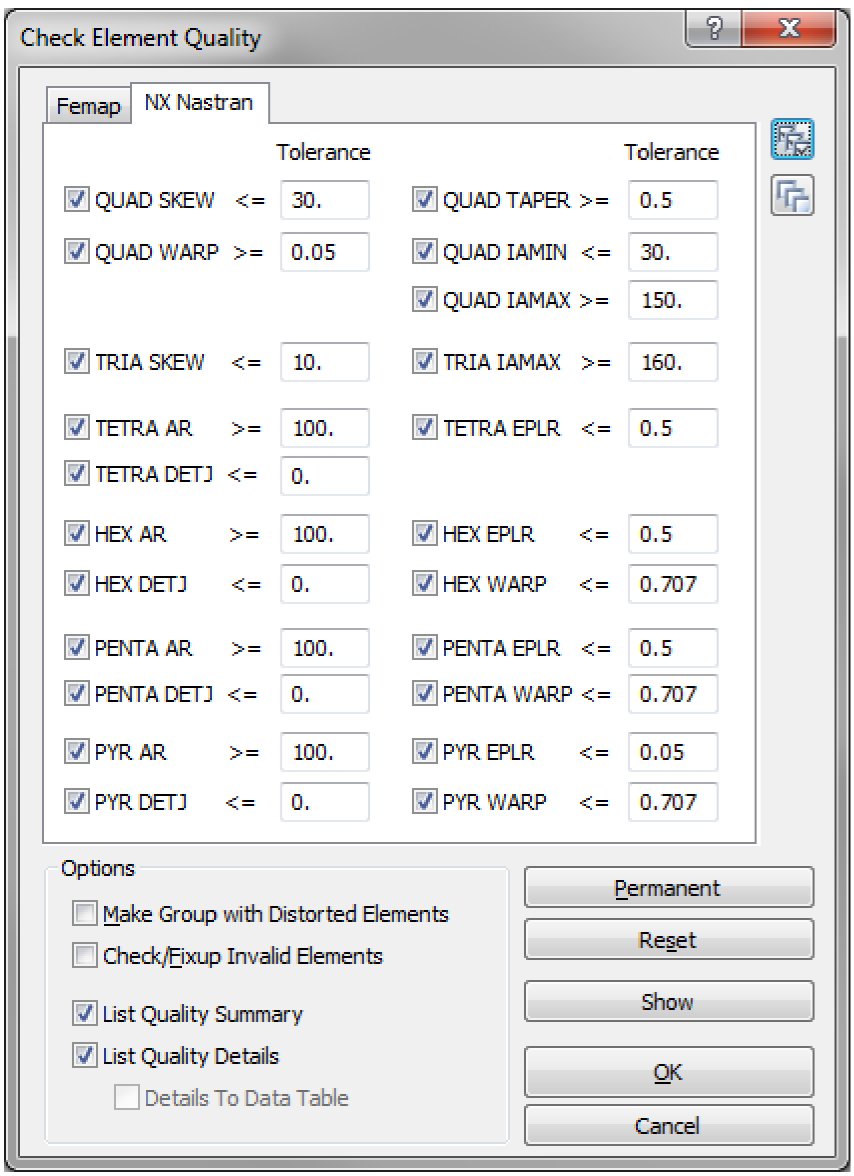
- Added *Old Version Import* option to potentially use old translator for legacy pre-v970 results files.
- Added support for *Translational Accel/Gravity* and *Rotational Velocity* body loads allowing them to be functionally-dependent.
- Added support for *Rate Effect - VP* for “24..LS-DYNA Piecewise Linear Plasticity” material type, must be a value between -1 and 1.
- Added support for *CST (0, 1 or 2)* and *SCOOR (-3 to 3)* for “66..LS-DYNA Linear Elastic Discrete Beam” material type.
- Added support to properly read d3plot files which contain SPH nodes.

A number of bugs were corrected.

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

Tools

- Added NX Nastran Element Quality checks to the *Tools, Check, Element Quality* command, which can be accessed by clicking the NX Nastran tab in the *Check Element Quality* dialog box.



There are 22 separate checks on the *NX Nastran* tab, which are the same quality checks used by the NX Nastran solver (WARP = Warping, IAMIN = Minimum Internal Angle in degrees, IAMAX = Maximum Internal Angle in degrees, AR = Aspect Ratio, EPLR = Edge Point Length Ratio, DETJ = Jacobian):

Quadrilaterals - QUAD SKEW, QUAD TAPER, QUAD WARP, QUAD IAMIN, QUAD IAMAX

Triangles - TRIA SKEW, TRIA IAMAX

Tetrahedrals - TETRA AR, TETRA EPLR, TETRA DETJ

Hexahedrals (Bricks) - HEX AR, HEX EPLR, HEX DETJ, HEX WARP,

Wedges - PENTA AR, PENTA EPLR, PENTA DETJ, PENTA WARP

Pyramids - PYR AR, PYR EPLR, PYR DETJ, and PYR WARP

Note: Because the element quality checks on the *NX Nastran* tab are the same used by the NX Nastran solver, please see the “Element Geometry Checks” section in chapter *15. Model Verification* of the *NX Nastran User’s Guide* for more information on how the various element quality checks are performed, as well as the entry for GEOMCHECK in the *NX Nastran Quick Reference Guide*.

- Updated *Tools, Measure, Distance Between Nodes* to return the “delta coordinates” between the nodes rather than the vector between the nodes, which makes it consistent with *Tools, Measure, Distance* and is more useful for non-rectangular coordinate systems.
- Updated the *Tools, Measure, Distance Between Geometry* command to allow you to choose *Nodes* in the *From* section, enabling you to be able to measure the distance between a node and geometric entities.
- Updated the *Tools, Mass Properties, Mesh Properties* command by adding the *Create Node at Total Center of Gravity* option to the *Check Mass Properties* dialog box.

Model Merge

- Added Add Related and Associated Entities button to Entity Selection section, which adds entities both referenced by other entity types currently in the *Entities to Merge* list and entities associated to those entities (i.e., mesh associated to geometry or vice versa). For instance, if *Entity Selection* is set to *Group* and the selected group only contains elements which have been sent to the *Entities to Merge* list, then pressing this button will add Node, Material, Property, and Layer entity types, and potentially other referenced entities, along with any associated geometric entities to the *Entities to Merge* list.

OLE/COM API

New and modified API Objects and Attributes

- Added Analysis Study (feAnalysisStudy) object to the API. Also, added Title, AnalysisProg, AnalysisType, FileTime, Notes, and AnalysisSet attributes to the Analysis Study Object.
- Added CaseLabel, ContactOption, ContactSetType, MSCNasCnlConvergenceFlags, and vMSCNasCnlConvergenceFlags attributes to the Analysis Case Object.
- Added NasExtSEOutAssignForm, NasMscCtOn, NasMscCtEnable, NasMscCtDDULMT, NasMscCtRVCNST, NasMscCtSLDLMT, NasMscCtTAUGMNT,NasMscCtAUGMENT, NasMscCtBEAMB, NasMscCtERRBAS, NasMscCtFTYPE, NasMscCtIBSEP, NasMscCtICSEP, NasMscCtMAXSEP, NasMscCtMETHOD, NasMscCtNLGLUE, NasMscCtNODSEP, NasMscCtSEGSYM, NasMscCtAUGDIST, NasMscCtBIAS, NasMscCtERROR, NasMscCtFNTOL, NasMscCtPENALT, NasMscCtSTKSLP, NasMscCtTPENALT, NasBulkMgrid, NasBulkSwpangle, NasBulkAlpha1, NasBulkAlpha2, NasBulkSwpangleVal, NasBulkMdofVal, NasBulkMgridID, NasBulkAlpha1Val, vNasBulkAlpha1Val, NasBulkAlpha2Val, vNasBulkAlpha2Val, NasAerobOPPHIPA, NasNXStratBeamalg, NasExecGPU, NasBulkEndTextOutsideBulk, CaseLabel, ContactOption, ContactSetType, MSCNasCnlConvergenceFlags, and vMSCNasCnlConvergenceFlags attributes to the Analysis Manager Object. Also, updated NasMCheckDataTol to be a real number instead of an integer and also updated vNasGCheckTest2, vNasGCheckTol2, and vNasGCheckMsg2 to allow you to control the PYR_DETJ option in GEOMCHECK.
- Added AxisMajorColor and AxisMajorColorAutoDim attributes to the Chart Object.
- Added PositionIsRelative attribute to the Chart Data Series Object.

- Added Midpoint, Center, GetPrecision, and ResetPrecision attributes to the Curve Object.
- Added ExtendEdgeValues attribute to the Data Surface Object.
- Added NastranQuadSkewOn, NastranQuadSkewLimit, NastranQuadTaperOn, NastranQuadTaperLimit,NastranQuadWarpOn, NastranQuadWarpLimit, NastranQuadIAMinOn, NastranQuadIAMinLimit, NastranQuadIAMaxOn, NastranQuadIAMaxLimit, NastranTriaSkewOn, NastranTriaSkewLimit, NastranTriaIAMaxOn, NastranTriaIAMaxLimit, NastranTetraAROn, NastranTetraARLimit, NastranTetraEPLROn, NastranTetraEPLRLimit, NastranTetraDetJOn, NastranTetraDetJLimit, NastranHexAROn, NastranHexARLimit, NastranHexEPLROn, NastranHexEPLRLimit, NastranHexDetJOn, NastranHexDetJLimit, NastranHexWarpOn, NastranHexWarpLimit, NastranPenAROn, NastranPenARLimit, NastranPenEPLROn, NastranPenEPLRLimit, NastranPenDetJOn, NastranPenDetJLimit, NastranPenWarpOn, NastranPenWarpLimit, NastranPyrAROn, NastranPyrARLimit, NastranPyrEPLROn, NastranPyrEPLRLimit, NastranPyrDetJOn, NastranPyrDetJLimit, NastranPyrWarpOn, and NastranPyrWarpLimit attributes to the Element Quality Object.
- Added SectionPlaneBase, vSectionPlaneBase, SectionPlaneNormal, vSectionPlaneNormal, SectionCurve, SectionVectorBase, vSectionVectorBase, SectionVectorTip, vSectionVectorTip, SectionRotateSums, SectionIncludeClippedElem, SectionTolerance, SectionSumLocation, SectionLimitToGroup, SectionGroup, ReverseTotalValues, SectionMode, SectionRadius, SectionReverse, SectionLocation, and SectionVectorTip attributes to the Freebody Object.
- Added InitAsInteger attribute to the Output Object.
- Added study and combination_type attributes to the Output Set Object.
- Added AssignForm attribute to the Superelement Reference Object.
- Added ComplexSyncMethod, ComplexSyncPhase, and ComplexSyncIncrement attributes to the View Object.

Removed API Objects and Attributes

- Removed type attribute from the Connection Object.
- Removed TetRecoveryMesher, TetPreV10TetMeshing, and TetMaxElemToAllocate attributes from to the Meshing Object.

New API Methods

- Added CountOutputSets, OutputSets, HasOutputSet, AddOutputSets, RemoveOutputSets, RemoveAllOutputSets and Delete to the Analysis Study object.
- Added AeroChordXYZ and AeroSpanXYZ to the Aero Panel object.
- Added AddNastranElementChecks to the Data Table object.
- Added SetPlateThickness to the Element object.
- Added GetNastranQuadSkew, NastranQuadSkew, GetNastranQuadTaper, NastranQuadTaper, GetNastranQuadWarp, NastranQuadWarp, GetNastranQuadIAMin, NastranQuadIAMin, GetNastranQuadIAMax, NastranQuadIAMax, GetNastranTriaSkew, NastranTriaSkew, GetNastranTriaIAMax, NastranTriaIAMax, GetNastranTetraAR, NastranTetraAR, GetNastranTetraEPLR, NastranTetraEPLR, GetNastranTetraDetJ, NastranTetraDetJ, GetNastranHexAR, NastranHexAR, GetNastranHexEPLR, NastranHexEPLR, GetNastranHexDetJ, NastranHexDetJ, GetNastranHexWarp, NastranHexWarp, GetNastranPenAR, NastranPenAR, GetNastranPenEPLR, NastranPenEPLR, GetNastranPenDetJ, NastranPenDetJ, GetNastranPenWarp, NastranPenWarp, GetNastranPyrAR, NastranPyrAR, GetNastranPyrEPLR, NastranPyrEPLR, GetNastranPyrWarp, NastranPyrWarp, GetNastranPyrDetJ, NastranPyrDetJ, and CheckNastranQuality to the Element Quality object.
- Added GetSectionCutSums and SetLocationFromCoord to the Freebody object.
- Added AreDuplicate to the Material object.
- Added PutCoordArray to the Node object.
- Added SetComplexOptions to the Output object.
- Added SetCombination, SetStudyCombination, ExpandCombination, and SetComplexOptions to the Output Set object.
- Added AreDuplicate to the Property object.

- Added ClearSearch to the ReadFile object.
- Added SetComplexOptions to the Results Browsing object.
- Added GetSelectedID and Tooltips to the Selector object.
- Added Show to the Set object.
- Added SheetFacesAsSet, WireCurvesAsSet, and Inside to the Solid object.

The following functions have been added or updated:

- feSurfaceNonManifoldAddExpress
- feCurveOffsetCurveWasher2
- feCurvePad
- feFileReadStepOpt2
- feFileWriteStep2
- feAppMessageClear
- feGetRealLength
- feModifySurfaceNormal
- feMeshEdgeSplit2
- feCurveProjectCurvesOntoSurfaces
- feSolidExtendEdgesToSurfaces
- feSolidRemoveRedundantPoint
- feSolidRemoveBlendsBelowRadius
- feSolidMovePointOntoGeometry
- feSolidRayFire
- feSurfaceRayFire
- feSolidExtendToSurface
- feSolidMidSurfaceManualInput
- feGroupsContaining
- feMeasureDistanceBetweenNodes2
- feMeshUnzip2
- feMeasureDistanceBetweenSolids (Corrected Spelling of function)
- feMeasureDistanceBetweenGeometry

The following functions have been removed

- feOutputTransform

New and updated Global Variables

- Added Pref_ElemQualQuadSkew, Pref_ElemQualQuadSkewVal, Pref_ElemQualQuadTaper, Pref_ElemQualQuadTaperVal, Pref_ElemQualQuadWarp, Pref_ElemQualQuadWarpVal, Pref_ElemQualQuadIAMin, Pref_ElemQualQuadIAMinVal, Pref_ElemQualQuadIAMax,Pref_ElemQualQuadIAMaxVal, Pref_ElemQualTriaSkew, Pref_ElemQualTriaSkewVal, Pref_ElemQualTriaIAMax, Pref_ElemQualTriaIAMaxVal, Pref_ElemQualTetAspectRatio, Pref_ElemQualTetAspectRatioVal,Pref_ElemQualTetEPLR, Pref_ElemQualTetEPLRVal, Pref_ElemQualTetDetJ, Pref_ElemQualTetDetJVal, Pref_ElemQualHexAspectRatio, Pref_ElemQualHexAspectRatioVal, Pref_ElemQualHexEPLR, Pref_ElemQualHexEPLRVal,Pref_ElemQualHexDetJ, Pref_ElemQualHexDetJVal, Pref_ElemQualHexWarp, Pref_ElemQualHexWarpVal, Pref_ElemQualPenAspectRatio, Pref_ElemQualPenAspectRatioVal, Pref_ElemQualPenEPLR, Pref_ElemQualPenEPLRVal,Pref_ElemQualPenDetJ, Pref_ElemQualPenDetJVal, Pref_ElemQualPenWarp, Pref_ElemQualPenWarpVal, Pref_ElemQualPyrAspectRatio, Pref_ElemQualPyrAspectRatioVal, Pref_ElemQualPyrEPLR, Pref_ElemQualPyrEPLRVal, Pref_ElemQualPyrWarp, Pref_ElemQualPyrWarpVal, Pref_ElemQualPyrDetJ, and Pref_ElemQualPyrDetJVal to set NX Nastran Element Quality values in the preferences.

- Added Pref_CreateResultStudy, Pref_ResultStudyTitle, Pref_TabPasteIgnoreDelimeter, Pref_SnapTo, Pref_RenderMaxMagnification, Pref_StudyAppendFemapTitles to set various preferences.
- Added Info_ViewShowNormal, Info_ViewShowTransparent, Info_ComplexSyncMethod, Info_ComplexSyncPhase, and Info_ComplexSyncIncrement.
- Added SolidAllowNonManifold to allow bodies to become NonManifold after geometry operations.
- Updated Pref_OutputSetTitles to set the Nastran Output Set Title preference.
- Updated Info_SnapTo, Info_SnapStyle, Info_MatlAngleDir, Info_ModelSizeX, vInfo_ModelSizeX, Info_ModelSizeY, vInfo_ModelSizeY, Info_ModelSizeZ, vInfo_ModelSizeZ.

Preferences

Graphics

- Added *Max Mag* to *Graphics Options* section

Specifies the maximum displayed magnification factor allowed by the graphics window. By default, the maximum magnification factor is 10,000. Once the magnification factor reaches the specified value, the level of magnification in the graphics window cannot be increased via zooming or by scrolling the mouse wheel. An error message will be issued to the *Messages* window when the specified limit has been reached.

- Updated what is supported by the Performance Graphics options in the *Graphics Options* section.

If this option is selected, a new architecture is used to improve graphics performance of both initial draw and dynamic rotation. This new architecture requires at least OpenGL 4.2 and uses the graphics processors to take on more of the graphics workload. This drastically reduces the amount of data that has to be transferred to the graphics card, which reduces transfer times and data creation times, along with removing the requirement to group like entities, which greatly improves performance for models with a very large number of properties (thousands).

Note: A command to quickly toggle *Performance Graphics* on/off may be added to any menu or toolbar using the *Tools, Toolbars, Customize* command. This command is located in the *Additional Commands* category on the *Commands* tab of the *Customize* dialog box.

When displaying nodal contours of nodal results in *Performance Graphics*, plate and solid element faces are only contoured if all nodes on that face have nodal results. This is a fundamentally different approach than what is done using the existing graphics architecture. The advantage of this approach is that contours are only drawn on the faces where nodal based output, such as contact results, actually exists, and not drawn on the entire element or on elements which do not have this type of output at all. Due to this different approach, a nodal contour of “sparse” nodal vector data, such as constraint force or moment, may not show any contours in *Performance Graphics*. It is recommended this data be examined by using either *Deformed Style* set to *Vector* or *Contour Style* set to *Vector*.

For the second full release of *Performance Graphics*, the following entities, styles, view options, and other capabilities are NOT supported by *Performance Graphics*:

Entities - Curves, Combined Curves, Surfaces (includes surfaces on solids and volumes), Boundary Surfaces (includes boundary surfaces on solids), Elements (Rigid - RBE1, RBE2, RBE3, and RSPLINE; Slide Line; Weld/Fastener), Connections (Connection Regions and Connectors), Other Regions (Fluid, NonStructural Mass, Bolt, Rotor), Loads (Geometry based Loads on Points, Curves, and Surfaces; Bolt Preload; Body Loads), Constraints (Geometry based Constraints on Points, Curves, and Surfaces; Constraint Equations), Aeroelasticity entities (Aero Panel/Body, Aero Spline, Aero Control Surface), and Text entities.

Note: When entity types and element types not supported by *Performance Graphics* exist in the model, they are drawn as they have been in previous versions with OpenGL.

Deformed Style - *Vector* and *Streamline*

Note: An error will be issued and nothing related to *Vector* or *Streamline* will be drawn.

Contour Style - *Beam Diagram*, *IsoSurface*, *Section Cut*, and *Vector.*

Note: When using *Beam Diagram*, beam diagrams are drawn using the same method used in older versions, while an error will be issued and nothing related to *IsoSurface*, *Section Cut*, or *Vector* is drawn.

View Options (Category - Labels, Entities and Color) - Any view option associated with an element type or entity type not currently supported by *Performance Graphics* is also not supported. In addition, *Element - Orientation/Shape* with *Element Shape* set to “3..Show Cross Section” (cross-sections drawn as they have been in previous versions with OpenGL) and *Constraint* when *Label Mode* is set to “4..Arrows, No Labels”, “5..Arrows, DOF”, “6..Triangles, No Labels”, or “7..Triangles, DOF” (only labels will be visible, no symbols).

View Options (Category - Tools and View Style) - *Axisymmetric Axes*, *View Legend*, *View Axis*, *Origin*, *Workplane and Rulers*, and *Workplane Grid* are simply drawn as they have been in previous versions with OpenGL.

View Options (Category - PostProcessing) - *Post Titles* and any view option associated with a Deformed Style, Contour Style, or PostProcessing Tool not currently supported by *Performance Graphics* is also not supported. In addition, *Contour/Criteria Style* when the *Filled (or Line)* option is not checked (changes appearance of *Contour/Criteria Legend*, but still shows “filled” contour).

Advanced Post - Commands on *View, Advanced Post* menu including *Dynamic Cutting Plane, Dynamic IsoSurface, Dynamic Streamline,* and *Beam Cross Section*.

Note: For *Contour Model Data*, the *Beam Diagram* option is not supported, but beam diagrams will be drawn using the same method used in older versions.

Entity Colors - *Pattern/Transparency* and *Line Style* options other than the default setting.

Entity Highlighting - *Window, Show Entities* command and various *Show When Selected* icon buttons

Individual Entity Transparency - *Modify, Transparency, Surface/Element/Property/Material*

Export of JT Files - cannot be written when *Performance Graphics* mode is enabled. Simply turn the *Performance Graphics* option off before using the *File, Picture, Save JT* command.

User Interface

- Added *Snap To* option to *Graphical Selection* section.

This option controls which Snap Mode will be used as the default Snap Mode when a model is opened. The available options are “0..Screen” (default),“1..Grid”, “2..Point”, “3..Node”, and “4..Smart”. See Section 4.2.3, “Quick Access Menu (Right Mouse Button)” in the FEMAP User Guide for more information on the various snap modes.

- Added *Ignore Delimiters if Pasting Tabs* option to *International Localization/Clipboard* section.

The *Ignore Delimiters if Pasting Tabs* option, on by default, allows the user to ignore delimiters, such as a comma (or period in certain regions), when pasting from the clipboard, provided there is a tab between the two values. For example, a spreadsheet contains values in 2 columns, each with a value containing a comma (Row 1, Column 1 = 1,001; Row 1, Column 2 = 5,050), and a tab between the two values when copied to the clipboard. When this option is ON, the values pasted into FEMAP from the clipboard are 1001 and 5050, while when this option is OFF, the values pasted into FEMAP are 1 and 1, while the 5,050 value is completely ignored.

Geometry/Model

- Updated the *Element Quality Preferences* dialog box accessed via the *Element Quality* button to be tabbed to allow you to enter default values for the FEMAP element quality checks on the FEMAP tab, while allowing you to enter default values for the NX Nastran element quality checks on the NX Nastran tab. Also, added the *All On* and *All Off* icon buttons to make it easier to turn on/off all quality checks on the current tab. Finally, added a *Restore Tab Defaults* button to restore the default values for the current tab.
- Updated the *Current Output Orientation* dialog box accessed via the *Output Orientation* button by allowing you to choose “3..Material Direction” from the drop-down control for *Force*, *Stress*, and *Strain* in the *Tria3*, *Tria6*, *Quad4*, and *Quad8* sections.
- Removed the *Pre-v11.1 Tet Meshing* option from the *Meshing and Properties* section.

Interfaces

- Removed the *Output Set Titles* drop-down control from the *Nastran Options* section, as it has been moved to the *File Options* section of the *Results* tab.

Results

- Added *Create Studies* option to *File Options* section.

When enabled, a Study will be created automatically when results are imported or attached to FEMAP. Each study will include all output data found in a particular results file (i.e., all subcases from a static analysis, all modes from a normal modes analysis, all frequencies or time steps from a dynamic analysis, all load steps from a nonlinear analysis, etc).

Note: Any number of output files may be selected during a single use of the *File, Attach to Results* command. If multiple output files are attached at one time, an individual Study will be created for each output file.

- Added *Study Titles* drop-down control to *File Options* section.

The option is used when the *Create Studies* option is enabled and is used to specify how automatically created Studies will be titled. The available options are explained in detail below:

Default - When importing or attaching to an OP2 or XDB file, it will try to use the Nastran Title, if it exists in the file, otherwise, it will use the File Name. For all other output files, it will use the File Name.

Analysis Set Title - This option is only intended for use when submitting jobs directly from FEMAP to a solver and will use the name of the Analysis Set currently being solved as the Study title. If importing or attaching to a results file, the Study title will correspond to the title of the Analysis Set currently “active” in the Analysis Set Manager, which may not be useful.

File Name - When this option is selected, the file name of the imported or attached to results file will be used as the Study title.

Nastran Title - When this option is selected, the TITLE from an imported OP2 file or attached to XDB file will be used as the Study title. For all other results files, “Analysis Study” will be used as the Study title.

- Added *Nastran Output Set Titles* drop-down control to *File Options* section (previously, this option was called *Output Set Titles* and was found in the *Nastran Options* section of the *Interfaces* tab).

When set to anything other than “0..Standard”, specifies which of the various Nastran labeling options will be used to name output sets created when importing output from or attaching to output in OP2 or XDB results files. Options available are TITLE, SUBTITLE, LABEL, and SUBTITLE, LABEL (appears as SUBTITLE - LABEL).

Note: This option is not available when importing results from the .f06 results file.

- Added *Append Femap Title* option to *File Options* section.

When this option is enabled, any value associated with an output set, such as a frequency or time, will be added to the end of the title of each output set during import or attach of the results file.

Note: When on, the associated value will always be added to the end of the output set title(s), therefore it is possible that longer output set title(s) may be truncated to accommodate the value. Typically truncation will only occur if the TITLE, SUBTITLE, and/or LABEL contain a relatively large number of characters and the option for *Nastran Output Set Titles* is set to something other than “0..Standard”.

- Removed the *Read Nonlinear Output* option from the Auto Answer General Post Read Questions section, as it is no longer needed due to FEMAP being able to import both the Linear and Nonlinear results from a nonlinear analysis into the same output set.