What's New in FEMAP 11.4

FEMAP 11.4.1 and 11.4.2 includes enhancements and new features, which are detailed below:

User Interface

Geometry Interfaces

Analysis Program Interfaces

Output and Post-Processing

OLE/COM API

FEMAP 11.4 includes enhancements and new features, which are detailed below:

User Interface

Geometry

Meshing

Elements

Materials

Loads and Constraints

Connections (Regions, Properties, and Connectors)

Listing

Groups and Layers

Views

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

Tools

OLE/COM API

Preferences

FEMAP 11.3.2 includes enhancements and new features, which are detailed below.

Geometry Interfaces

Element Update

Analysis Program Interfaces

OLE/COM API

FEMAP 11.3.1 includes enhancements and new features, which are detailed below.

User Interface

Meshing

Mesh Associativity

Connections (Regions, Properties, and Connectors)

Loads and Constraints

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

OLE/COM API

FEMAP 11.3 includes 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

What's New for version 11.4.1 and 11.4.2

User Interface

Menu, Toolbars

Menu

- Added *Tools, TMG Thermal/Flow Analysis* command to the *Tools* menu for FEMAP 11.4.2.
- Added *Help, TMG Thermal and Flow* command to the *Help* menu for FEMAP 11.4.2.

Toolbars

- Added *TMG Thermal/Flow* icon to *Panes* toolbar for 11.4.2. This icon is used to launch the *Thermal/Flow Model Info* pane. For more information on the use of the *Thermal/Flow Model Info* pane, use the *Help, TMG Thermal and Flow* command to launch a HTML version of the *Femap Thermal/Flow User Guide*.

Geometry Interfaces

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

FEMAP Interface	Latest Supported Version
Parasolid	Parasolid 30.0
Solid Edge	Solid Edge with Synchronous Technology 10
Pro/Engineer	Creo 4
NX	NX 12.0 - updated for FEMAP 11.4.2

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:

- NX Nastran Interface

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

NX Nastran Interface

- FEMAP with NX Nastran bundle now includes NX Nastran 11.0.2.
- Version 11.4.2 now supports NX Nastran 12.0.

Output and Post-Processing

- Added ability to transform magnitude/phase complex output data.

OLE/COM API

New and Updated API Methods

- Added GetSavedSetType to the Set Object.

- Added CollectorSingleSymbol, CollectorAddSymbolEntityFaceNormalAutoLocations, CollectorAddSymbolEntityEdgeLocations, CollectorAddSymbolEntityEdgeAutoLocations, CollectorAddSymbolREAL8EntityFaceNormalAutoLocations, CollectorAddSymbolREAL8EntityEdgeLocations, and CollectorAddSymbolREAL8EntityEdgeAutoLocations to the User Defined Graphics Object.
- Updated SetTri3Orientation, SetTri6Orientation, SetQuad4Orientation, and SetQuad8Orientation for the Results Browsing Object, by allowing user to specify Material Direction for each argument. In addition, updated SetSolidOrientation to allow user to specify Element for each argument.

The following functions have been added or updated:

- feAppUndoClear
- feChartPaneCopy
- feEntitySetVisibility2

New and updated Global Variables

- Added UndoGrayed global variable.
- Updated Info_OrientTria3StressOuput, Info_OrientTria3StrainOuput, Info_OrientTria3ForceOuput, Info_OrientTria6StressOuput, Info_OrientTria6StrainOuput, Info_OrientTria6ForceOuput, Info_OrientQuad4StressOuput, Info_OrientQuad4StrainOuput, Info_OrientQuad4ForceOuput, Info_OrientQuad8StressOuput, Info_OrientQuad8StrainOuput, and Info_OrientQuad8ForceOuput global variables, by allowing user to specify a value of 3 to specify Material Direction. In addition, Info_OrientSolidIsoOutput, Info_OrientSolidAnisoOutput, and Info_OrientSolidHyperOutput now allow user to specify a 3 for Element.
- Updated Pref_OrientTria3StressOuput, Pref_OrientTria3StrainOuput, Pref_OrientTria3ForceOuput, Pref_OrientTria6StressOuput, Pref_OrientTria6StrainOuput, Pref_OrientTria6ForceOuput, Pref_OrientQuad4StressOuput, Pref_OrientQuad4StrainOuput, Pref_OrientQuad4ForceOuput, Pref_OrientQuad8StressOuput, Pref_OrientQuad8StrainOuput, and Pref_OrientQuad8ForceOuput preference variables, by allowing user to specify a value of 3 to specify Material Direction. In addition, Pref_OrientSolidIsoOutput, Pref_OrientSolidAnisoOutput, and Pref_OrientSolidHyperOutput now allow user to specify a 3 for Element.

New and updated Events

- Added FEVENT_UPDATEDSAVEDSET event, which indicates a “SavedSet” Set Object has been updated.
- Added FEVENT_LEAVINGMODEL event, which can be used in conjunction with FEVENT_SWITCHMODEL and indicates about to leave a model. lParam contains the identifier of the model being left.
- Added FEVENT_CLOSINGMODEL event, which can be used in conjunction with FEVENT_ENDMODEL and indicates closing a model. lParam contains the identifier of the model being closed.

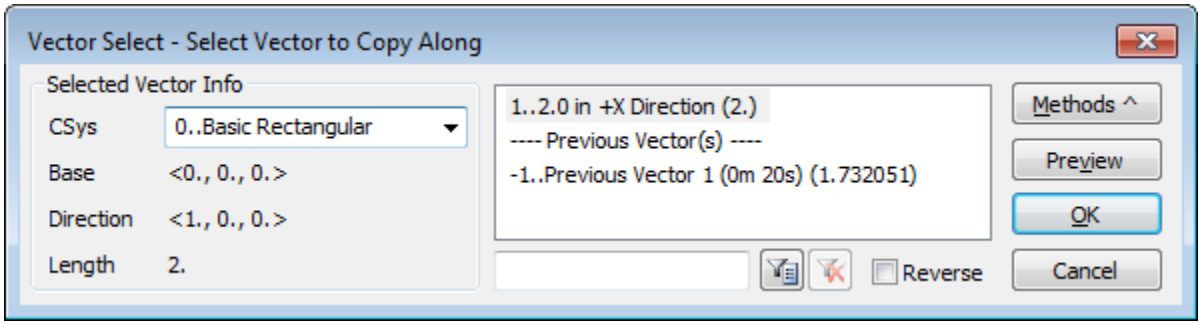
What's New for version 11.4

User Interface

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

General

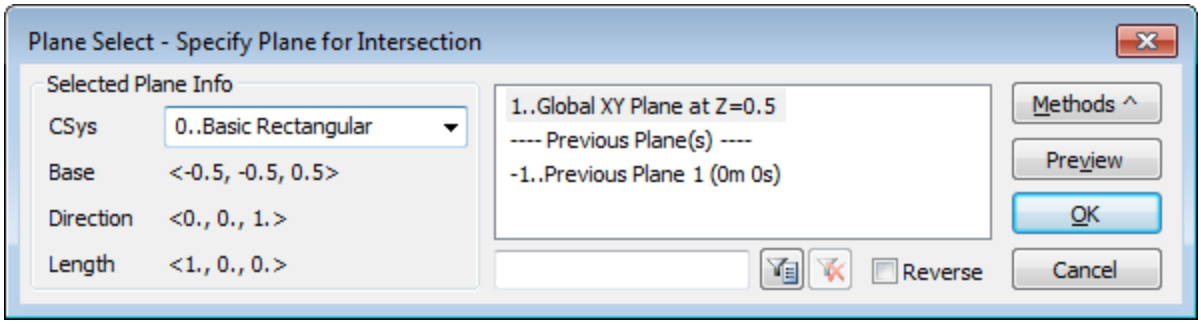
- Added *Tetra EPIA*, *Pyr EPIA*, *Penta EPIA*, and *Hex EPIA* as options when using the *Model Data Value* option for *Pick^* in the standard entity selection dialog box.
- Added *Saved* option to the *Methods^* menu on the standard *Vector* definition dialog box.



This method allows selection of a vector which has been saved to the database using the *Tools, Vector Manager* command (Section 7.1.3, "Tools, Vector Manager") or from the 5 vectors most recently defined during a command. Once a vector is highlighted in the list, a preview of the vector will be displayed in the graphics window. In addition, the *Selected Vector Info* section displays the *Base* and *Direction* of the highlighted vector in the coordinate system currently selected in the *CSys* drop-down, along with the *Length*. Finally, The *Reverse* option can be used to reverse the direction of the highlighted vector, while maintaining the *Length*.

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

- Added *Saved* option to the *Methods^* menu on the standard *Plane* definition dialog box.



This method allows selection of a plane which has been saved to the database using the *Tools, Plane Manager* command (Section 7.1.3, "Tools, Vector Manager") or from the 5 planes most recently defined during a command. Once a plane is highlighted in the list, a preview of the plane will be displayed in the graphics window. In addition, the *Selected Plane Info* section displays the *Base* location, *Normal* vector, and *Axis* vector of the highlighted plane in the coordinate system currently selected in the *CSys* drop-down. Finally, The *Reverse* option can be used to reverse the direction of the highlighted plane's normal vector.

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

Menu

- Added *Tools, Vector Manager...*, *Tools, Plane Manager...*, and *Tools, Function/Table Editor* to the *Tools* menu. See Tools section for more information.
- Added *Modify, Break* submenu, which contains two commands, *Modify, Break, At Location...* (previously *Modify, Break...* command) and *Modify, Break, At All Intersections*, to the *Modify* menu. See Geometry section for more information.
- Added *Help, Toolbars, Draw/Erase* and *Help, Dockable Panes, Function/Table Editor* to the *Help* menu.

Toolbars

- Added *Draw Mode Select From All* to the *Select Area* icon menu of the *Draw/Erase* toolbar. When this option is enabled, which is the default, all entities in the model which are not hidden by other visibility methods will be shown when selecting which entities to “Draw”. When disabled, entities which are currently being “Drawn” will remain the only visible entities and graphical selection only considers those entities.
- Updated *Load Group* command on *Select* toolbar to automatically turn on “Select Multiple” switch. Also, if the icon at the top of the drop-down in the toolbar is clicked, the *Load Group* command will be used again, after being used once. Also, updated *Grow* command to honor the “Select Related” switch, however if you *Grow* with “Select Related” enabled, *Shrink* will no longer remove entities from the selection.

Model Info tree

- Added *Autoscale to Show Entities* option to *Show When Selected* icon menu. When on, the active view (or all views if the *All Views* option is enabled) will be automatically centered and magnified around the highlighted entities.

Meshing Toolbox

- Added functionality which will automatically update “rigid spider” elements (RBE2 and RBE3) when dynamically updating the mesh with the *Meshing Toolbox*. This works best when all the nodes on a curve or a surface being updated are used by the rigid element. Otherwise, a message stating “Potential rigid disconnect at Element # and Curve #” or “Potential rigid disconnect at Element # and Surface #” may appear.

PostProcessing Toolbox

- Added *Reverse Freebody Values* option to *Freebody Tool*. See Output and Post-Processing for more information.

Charting pane

- Added *Complex Plot* drop-down to the *Chart Settings* tab of the *Charting* dialog box. In addition, added the ability to set certain options for the “Y Complex” axis. See Tools section for details.
- Updated *Chart Data Series* dialog box to use the *Type* drop-down to select the type of *Data Series* to create. The five types are “0..Vector vs Entity”, “1..Vector vs. Output Set”, “2..Vector vs. Vector”, “3..Expand Complex” (New for 11.4!), and “4..Function”. Also, the Data for each type of Data Series is now entered on the *Data* tab, which also contains new options to allow the user to *Transform*, *Convert*, and/or expand *Complex* output when plotted. Meanwhile, the settings for *Labels*, *Markers*, and *Color* are now on the *Style* tab, which is the same for all Data Series types. See Tools section for full description of the updates.

Entity Editor

- Added *Nastran EPIA* Element Quality check for parabolic tetrahedral, pyramid, wedge, and hexahedral element topologies.

Data Surface Editor

- Added *Data Conversion* drop-down to the *Define Options for Variation* dialog box, which is used to define the Data Conversion method for an *Output Map Data Surface*.

Specifies the method of data conversion for the selected output vector. To use the output as it appears contoured in the active view of the current model, use “0..From View”. The other options for data conversion, “1..Average”, “2..Maximum Value”, “3..Minimum Value”, “4..Average, No Corners”, “5..Max, No Corners”, and “6..Min, No Corners” correspond to options available on the *Contour Options* dialog box. For more information, see “Data Conversion” in the “Contour Options...” portion of Section 8.2.2, “Choosing Deformed and Contour Styles”.

Function/Table Editor

- The *Function/Table Editor* dockable pane was added for 11.4. See Tools section for full description.

Connection Editor

- Added *Autoscale to Show Entities* option to *Show When Selected* icon menu. When on, the active view (or all views if the *All Views* option is enabled) will be automatically centered and magnified around the highlighted entities.

Data Table

- Added *Autoscale to Show Entities* option to *Show When Selected* icon menu. When on, the active view (or all views if the *All Views* option is enabled) will be automatically centered and magnified around the highlighted entities.
- Added *Tetra EPIA*, *Pyr EPIA*, *Penta EPIA*, and *Hex EPIA* columns for parabolic tetrahedral, pyramid, wedge, and hexahedral element topologies, when using the *Add Nastran Element Checks...* command on the *Show/Hide Columns* icon menu.

Geometry

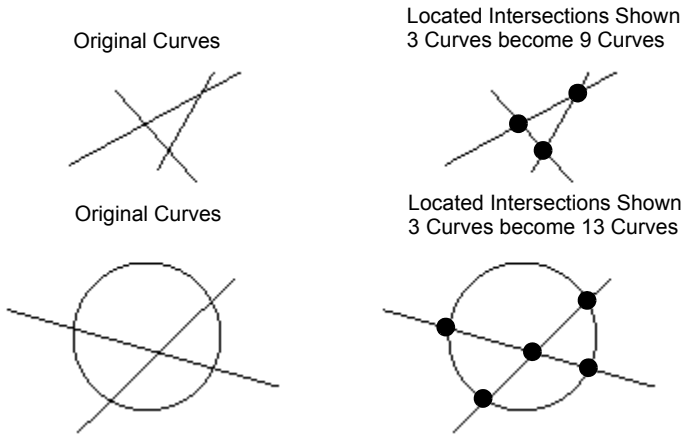
- Added *Modify, Break, At All Intersections* command.

... splits one or more curves into multiple curves based on all intersections found between the selected curves.

Simply select any number of curves using the standard entity selection dialog box, then the command will automatically locate all of the intersections between those selected curves and split each curve as many times as needed.

Normally, breaking a curve does not change its type. You just end up with multiple new curves of the same type, that together, make up the original curve. The only exception is when a circle is broken. In this case, you end up with two or more arcs (a different type of curve) that represent the original circle.

Example



- Updated *Modify, Update Other, Surface Normal* to work with Parasolid General Bodies.

Meshing

- Updated the *Color and Layer* and *Mesh Sizes, Loads, Constraints...* options in *Generate Options* dialog box to *Color, Layer, Formulations...* and *Loads, Constraints, Regions...*, respectively, which better describes what these options include when used by the *Mesh, Copy/Radial Copy/Scale/Rotate/Reflect...* commands.
- Updated the *Mesh, Editing, Element Refine* command to automatically update the element references in all groups, including any group which contains elements based on rules.
- Updated *Mesh, Reflect, Elements* to reflect any material orientations which are specified using an angle or vector (but not Matl CSys) when reflecting planar elements.
- Improved meshing of surfaces which are “very nearly planar segments” of cylinders or spheres to insure the nodes always lie on the surface.

Elements

- Updated the *Modify, Update Elements, Property ID* command to only show properties of the valid type if only elements of a single element type are selected to update.

Materials

- Added the *Electrical/Optical* tab for *Anisotropic (2D)* and *Anisotropic (3D)* material types.
- Updated the calculation used for conversion when changing the *Type* of material from *Anisotropic (3D)* to *Anisotropic (2D)*.

Loads and Constraints

- Updated the *Model, Load, Heat Transfer* command by adding an option to enable *View Factor Zero Tolerance* and enter a corresponding value to the *Radiation* section of the *Heat Transfer Loads* dialog box.
- Updated *Model, Load, Map Output From Model* command by adding *Data Conversion* drop-down to the *Map from Model Output* dialog box, which specifies the method of data conversion for the selected output vector. To use the output as it appears contoured in the active view of the current model, use “0..From View”. The other options for data conversion, “1..Average”, “2..Maximum Value”, “3..Minimum Value”, “4..Average, No Corners”, “5..Max, No Corners”, and “6..Min, No Corners” correspond to options available on the *Contour Options* dialog box. For more information, see “Data Conversion” in the “Contour Options...” portion of Section 8.2.2, “Choosing Deformed and Contour Styles”.
- Updated the *Model, Load, Combine* command by removing the *From Data Surface* option from the *Combine To* section and the *Data Surface* drop-down from the *Options* section. As an alternative, use a *Load Set Combination Table* in the *Function/Table Editor* to create more complicated combinations of Load Sets.

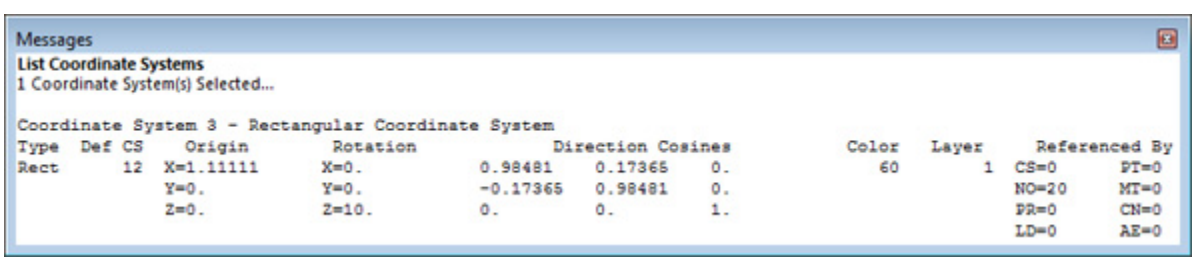
Connections (Regions, Properties, and Connectors)

- Added *Constrained* option and *Formul. Opt*. drop-down to the General section of the *LS-Dyna* tab in the *Define Connection Property* dialog box.

Listing

- Updated the *List, Model, Coord Sys* command to include listing the 3 X 3 Direction Cosines matrix relative to either the definition or listing coordinate system.

The coordinate system listing:



```
Messages
List Coordinate Systems
1 Coordinate System(s) Selected...

Coordinate System 3 - Rectangular Coordinate System
Type  Def CS    Origin        Rotation            Direction Cosines          Color   Layer     Referenced By
Rect      12  X=1.11111      X=0.          0.98481    0.17365    0.             60       1  CS=0      PT=0
              Y=0.           Y=0.          -0.17365   0.98481    0.                         NO=20     MT=0
              Z=0.           Z=10.         0.         0.         1.                         PR=0      CN=0
                                                                                             LD=0      AE=0
```

Groups and Layers

- Updated the *Mesh, Editing, Element Refine* command to automatically update the element references in all groups, including any group which contains elements based on rules.

Views

- Added *Autoscale* option to *Window, Show Entities* command. When on, the active view (or all views if the *All Views* option is enabled) will be automatically centered and magnified around the highlighted entities.

Output and Post-Processing

- Added *Reverse Freebody Values* option to the *Freebody Tool* in the *PostProcessing Toolbox*.

Reverse Freebody Values - When this option is disabled, which is the default, freebody entities use the grid point force values, as calculated by Nastran, to calculate forces and moments on each node, based on the selected contributions. When selecting elements to include on the contribution, FEMAP displays results for contributions from elements surrounding the elements that have been selected, which in turn, gives the forces and moments applied to the selected nodes and elements.

When this option is enabled, the sign on all Nastran grid point force output is reversed, so that the user can obtain an identical freebody calculation by including contributions from the selected freebody elements, instead of the surrounding peripheral elements. This is useful in situations where the model contains only elements that may be used for freebody selection, but not the elements that surround the area of interest (i.e, “carve-out models”)

Note: It is important to realize that it is still up to the individual analyst to ascertain which contributions are relevant to the type of analysis being performed, thus, this option should only be enabled when the analyst can confirm it is appropriate.

- Added *Show* icon button to *Freebody Tool* in the *PostProcessing Toolbox* to highlight, in the graphics window, both the nodes and elements used by the Freebody currently loaded in the *Freebody Tool*.
- Added ability to include beam section stresses currently being displayed by the *View, Advanced Post, Beam Cross Section* command, when creating a JT File.
- Updated *View, Advanced Post, Contour Model Data* command to allow display of *Tetra EPIA*, *Pyr EPIA*, *Penta EPIA*, and *Hex EPIA* element quality on parabolic solid elements.

Geometry Interfaces

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

FEMAP Interface	Latest Supported Version
Solid Edge	Solid Edge with Synchronous Technology 9
CATIA	CATIA V5-6R2016 SP2
SolidWorks	SolidWorks 2017

- Removed the “legacy” version of the CATIA V5 geometry translator, which could only be used once a “type-in” preference was specified in the [User] section of the FEMAP.INI file.
- Removed the “legacy” version of the SolidWorks geometry translator, which could only be used once a “type-in” preference was specified in the [User] section of the FEMAP.INI file.

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)

- ANSYS Interface
- ABAQUS Interface
- DYNA Interface

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

FEMAP Neutral File Interface

- Removed *Significant Digits* option from the *File Format* section of the *Neutral File Write Options* dialog box. All Neutral files are now written using “Max Precision”, which is 16 digits for double-precision real values, such as nodal coordinates, and 8 digits for single-precision real values, such as results.
- Updated Neutral Read and Write for v11.4 changes

NX Nastran Interface

- Added *Tetra EPIA*, *Pyr EPIA*, *Penta EPIA*, and *Hex EPIA* to *NASTRAN GEOMCHECK* dialog box, which will write the appropriate GEOMCHECK entries in Solution Control section of the input file.

A number of bugs were corrected

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

Nastran Interfaces (NX and MSC/MD)

- Added *Fiber* and *Curvature* options for *Strain* in the NASTRAN Output Requests dialog box. When using *Fiber*, which is the default, FIBER is written to the STRAIN entry in Case Control. When using *Curvature*, no additional text is written to the STRAIN entry, which was not possible before via the user interface.
- Added read and write support of the ZTOL field on the VIEW3D entry.
- Improved performance significantly when importing input files with a large number of DMIG entries.

ANSYS Interface

A number of bugs were corrected.

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

ABAQUS Interface

- Added support to attach to *.ODB files from ABAQUS version 2016.

A number of bugs were corrected.

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

DYNA Interface

- Added support for *CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR, *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR, *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED, and *CONTACT_AUTOMATIC_NODES_TO_SURFACE_SMOOTH with options specified via the *Formul. Opt.* drop-down in the *General* section of the *LS-DYNA* tab of the *Define Connection Property* dialog box. In addition, *CONTACT_FORMING_SURFACE_TO_SURFACE_MORTAR can also be specified by setting *Type* to “6..Forming” and using the *Formul. Opt.* drop-down.

A number of bugs were corrected

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

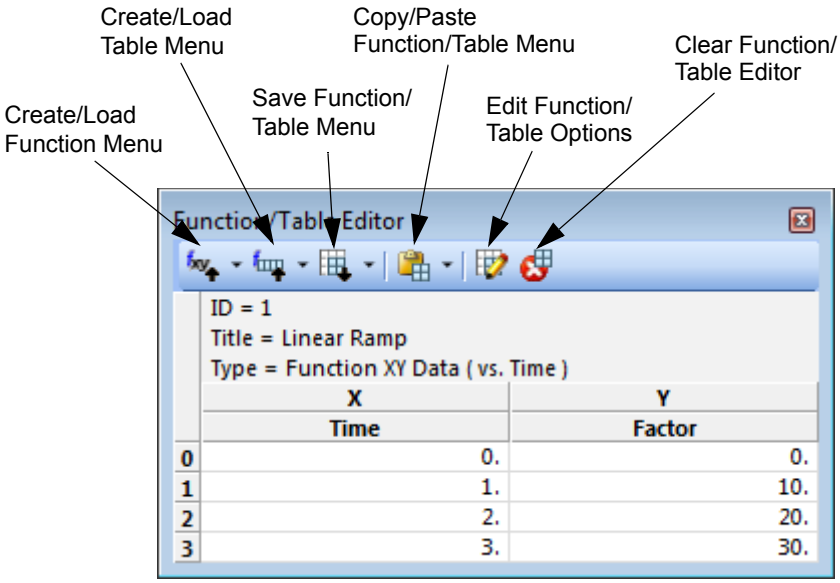
Tools

- Added the *Tools, Function/Table Editor* command.

The *Function/Table Editor* dockable pane allows you to create and edit Functions and Tables. A function always has two columns (XY Data), while the number of columns in a table depends upon the Type of table.

Functions allow you to input tables of parameters that are used to describe loading, material, or property behavior. and can also be created using the *Model, Function* command. For more information on how functions are used, see Section 4.7, “Working with Functions”. To create a new function using the *Function/Table Editor*, select a Type of function or the *Load Function From Library* command from the *Create/Load Function* menu.

Tables are similar to functions, as they also contain values in rows and columns, but a table always have more than two columns. To create a new table using the *Function/Table Editor*, select a Type of table from the *Create/Load Table* menu. For each unique Type of table found under *Thermal/Flow Vector Tables*, the number of columns depends on the selected Type. For the other process-specific tables, *Load Set Combination Table* and *Result Set Processing Table*, the total number of columns depends on the number of load sets or output sets the user selects, respectively.



When creating a new function or a new table from the types found under *Thermal/Flow Vector Tables*, you can enter a *Title*, select a different *Type*, if needed, and/or enter a value for *Initial Number of Rows*. When creating a *Load Set Combination Table* or *Result Set Processing Table* the same attributes may be specified, along with some additional options.

Once a function/table has been created using one of the methods, a corresponding table with the appropriate rows and columns will appear in the *Function/Table Editor* dockable pane. Now, the values can be added or modified in the table itself. You can also copy and paste individual cells or use the various commands available on the context-sensitive menu available when at least one cell is highlighted to populate the cell(s). Once a function/table has been completed, it can be saved to various places using the commands on the *Save Function/Table Menu*.

Function/Table Editor Icons

Create/Load Function menu

This icon menu contains various commands. The first section allows you to select a Type of function to create. The second section allows you to “reload” a single function into the *Function/Table Editor* or a delete any number of functions. The third section allows you to load a function into the *Function/Table Editor* from a library of previously saved functions.

Hint: It is very important to identify the proper type for the function that you are trying to define; otherwise, it will not be properly used when you try to analyze your model. Also, each solver is different, so one solver may use a certain type of function to model a certain behavior, while a different solve may require the use of a different type of function to model the same behavior. Please consult your particular solver’s documentation for more information.

Function Types - The types of functions are separated into 6 categories:

vs Time - a single type of function used to specify time-varying input for transient analysis

Temperature - this group of functions contains types used to define temperature-dependent behavior, typically for material properties, and includes *vs. Temperature* and *Function vs. Temperature* (*Function* refers to IDs of other stress/strain functions), as well as *NASTRAN TABLEM1 Linear* (X and Y Axes both use Linear interpolation), *NASTRAN TABLEM1 Log* (X and Y Axes both use Logarithmic interpolation), *NASTRAN TABLEM1 Linear, Log* (X Axis uses Linear interpolation, Y Axis using Logarithmic), and *NASTRAN TABLEM1 Log, Linear* (X Axis uses Logarithmic interpolation, Y Axis using Linear).

Stress/Strain - this group of functions contains types used to define nonlinear behavior, typically for material properties, and includes *Stress vs. Strain*, *Stress vs. Plastic Strain*, *vs. Strain Rate*, *Function vs. Strain Rate* (*Function* refers to IDs of other stress/strain functions), and *Strain vs. Stress*.

Frequency Dependent/Dynamics - *vs. Frequency* is used to specify frequency-varying input; *Structural Damping*, *Critical Damping*, and *Q Damping* are used to define modal damping tables for frequency response analysis; *Critical Damping vs. Function* (*Function* refers to IDs of other functions created as output from a response spectrum generation analysis) is used to specify input for a response spectrum application analysis; and *Mach Number* and *vs. Aerodynamic Factor* are used in aeroelastic analysis.

Thermal/Flow - this group of functions is used to specify solver-specific input for future versions of FEMAP Flow, FEMAP Thermal, and/or FEMAP Advanced Thermal. It is not currently possible to define these types of functions.

Other - this group contains function types which do not belong in any of the above categories. *Dimensionless* represents generic data in X and Y columns; *Acceleration vs. Location* should only be used when defining a "Varying Translational Acceleration" body load (see Section 4.3.3.1, "Model, Load, Body"); *vs. Curve Length* and *vs. Curve Param* should only be used to create variable loading conditions on curves (see Section 4.3.4.2, "Model, Load, On Curve..."); and the *Results Processing Only* functions are special functions created by the *Model, Output, Forced Response* command (see Section 8.5.14, "Model, Output, Forced Response...") and creating these type of functions using the *Function/Table Editor* is not advisable.

Reload/Edit Function... - Allows you to choose an existing function from a single selection dialog box, then reloads the attributes and data of the selected function into the *Function/Table Editor*, where the function may be viewed and/or edited, then saved again.

Delete Function... - Allows you to choose any number of existing functions in a multi-select dialog box, then deletes the selected functions.

Load Function From Library... - Allows you to choose a function from a FEMAP Function Library, then loads the attributes and data of the selected function into the *Function/Table Editor*, where the function may be viewed and/or edited, then saved. For more information on libraries, see Section 2.6.2.10, "Library/Startup" and Section 4.3.6, "Library Selection" of the FEMAP *User's Guide*.

Create/Load Table menu

This icon menu contains various commands. The first section allows you to select a Type of table to create. The second section allows you to "reload" a single table into the *Function/Table Editor* or a delete any number of tables.

Load Set Combination Table... - This type of table has one very specific use, to create new load sets based on existing load sets. Each selected existing load set will appear in the *Function/Table Editor* as an individual column, while each new load set to create will appear as a row.

Result Set Processing Table... - This type of table 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 table. These output sets are created or updated using the "Linear Combination", "RSS Combination", or "Envelope" methods also available in the *Model, Output, Process* command.

Thermal/Flow Vector Tables - this group of tables is used to specify solver-specific input for future versions of FEMAP Flow, FEMAP Thermal, and/or FEMAP Advanced Thermal. It is not currently possible to define these types of tables.

Reload/Edit Table... - Allows you to choose an existing table from a single selection dialog box, then reloads the attributes and data of the selected table into the *Function/Table Editor*, where the table may be viewed and/or edited, then saved again.

Delete Table... - Allows you to choose any number of existing tables in a multi-select dialog box, then deletes the selected tables.

Save Function/Table to Model Database menu

This icon menu contains various commands used to save the function/table currently loaded into the Function/Table Editor using several different methods.

Save... - simply saves the function/table with the current title to the FEMAP model

Save As... - allows you to specify an *ID* and update the *Title* of the current function/table when saving, using the *Create or Update* dialog box.

To File... - allows you to save the function/table values to a comma-separated file (*.CSV file).

Save Function to Library... - simply saves the function/table to the FEMAP function library.

Copy Text to Clipboard menu

This icon menu contains commands that allow the *Function/Table Editor* to interact with the clipboard.

Copy - copies the all text, including header information (*ID*, *Title*, *Type*, *Column Titles*, and *Row IDs*), to the clipboard in Rich Text Format (RTF) so it can be pasted into a different application.

Copy Data - copies the data from all editable cells to the clipboard

Paste Data - pastes data the *Function/Table Editor* has placed on the clipboard into the *Function/Table Editor*. This would typically be used to copy data from one function/table to create a new function/table in the Function/Table Editor, make changes, then save the function/table.

Edit Table and Clear Function/Table Icons

Edit Table Options - opens a dialog box that corresponds to the type of function/table currently in the *Function/Table Editor*. This allows you to modify the *Title* and *Options* for all functions/tables, as well as the *Type* for most functions/tables. If a *Load Set Combination Table* or *Result Set Processing Table* is loaded in the *Function/Table Editor*, the *Type* cannot be changed, as these tables possess options which are unique to these table types.

Clear Function/Table - completely clears any table and values currently in the *Function/Table Editor*.

Create Combined Load Sets/Create Processed Results Sets Icon

When a *Load Set Combination Table* or *Result Set Processing Table* is loaded into the *Function/Table Editor*, an icon will appear to the right of the *Clear Function/Table* icon. Once the table has been populated with the desired data, this icon can be used to the *Create Combined Load Sets* or *Create Processed Load Sets*, respectively.

Context Sensitive Menus (All Functions/Tables)

There is a context sensitive menu which will appear when cell(s) are highlighted in the *Function/Table Editor* pane and the right mouse button is clicked. This menu allows you to:

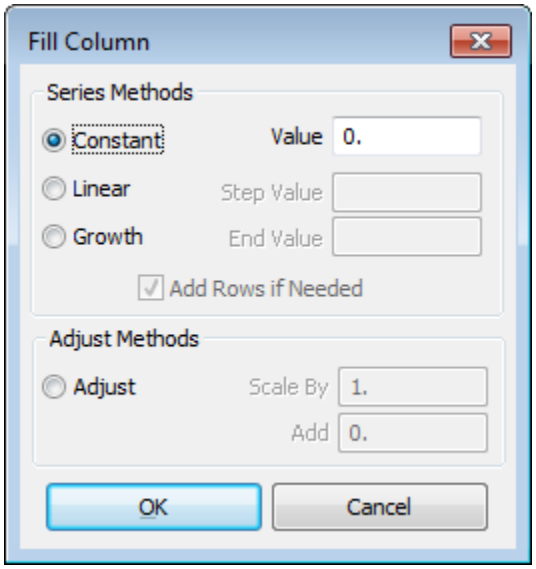
Copy Selected - copies the values in the highlighted group of cells to the clipboard.

Paste - pastes values from the clipboard into the highlighted cells.

Fill Down - copies the value at the “top” of the highlighted group of cells and “fills” all of the highlighted cells in the same column with that value. If multiple columns are highlighted, the value at the “top” of the cells in each column will be the value that is used.

Fill Right - similar to *Fill Down*, only it “fills” the highlighted cells in the same row with the value furthest to the left in the group of highlighted cells. If multiple rows are highlighted, the value furthest to the left of the highlighted cells in each row will be the value that is used.

Series... - displays the *Fill Column* dialog box, which contains options used to create a new series of values, starting with the highlighted cell, or adjust the existing value in each highlighted cell.



Series Methods

Constant - if highlighted cells are empty, the value specified in *Value* is entered into all highlighted cells. If the highlighted cells already have values, then the values in the cells of the column furthest to the left are modified.

Linear - creates a series of values starting at *Value* and ending at or below the specified *End Value*, incremented by the *Step Value*. If the highlighted cells are across multiple columns and the cells are empty, then the same series of values will be sent to each column. If the value in the "top-most" cell of each column is not empty, then that value will be used as the starting value of the series of values for each column. If *Add Rows if Needed* is enabled, which is the default, then rows will be automatically added to accommodate the entire series.

For example, *Value* = 100, *Step Value* = 5, *End Value* = 150, produces a series of values containing the following: 100, 105, 110, 115, 120, 125, 130, 135, 140, 145, and 150. If *End Value* is instead set to 149, then the last value in the created series would be 145 instead of 150.

Growth - creates a series of values starting at *Value* and ending at or below the specified *End Value*, with each subsequent value being the product of the previous value multiplied by the *Step Value*. If the highlighted cells are across multiple columns and the cells are empty, then the same series of values will be sent to each column. If the value in the "top-most" cell of each column is not empty, then that value will be used as the starting value of the series of values for each column. If *Add Rows if Needed* is enabled, which is the default, then rows will be automatically added to accommodate the entire series.

For example, *Value* = 1, *Step Value* = 2, *End Value* = 16, produces a series of values containing the following: 1, 2, 4, 8, 16. If *End Value* is instead set to 15, then the last value in the created series would be 8 instead of 16.

Adjust Methods

Adjust - modifies the values in highlighted cells by first multiplying the values using the value specified for *Scale By*, then adding the value specified for *Add* to the scaled values. If *Scale By* is set to 1.0, then only the value specified for *Add* will be used to modify the values.

Hint: Once cells already have values, the easiest process to assign a single, constant value to all of the highlighted cells is to set *Scale By* to 0.0, then set *Add* to the value you want to assign to those cells.

Clear Selected - clears the value(s) in the highlighted cell(s).

Sort Table - for functions, sorts both columns using the values in the *X* column, while for tables, sorts all columns using the values in the *Time* column. For both functions and tables, the lowest value will always be on the top, while the highest value will be at the bottom. Not available when a *Load Set Combination Table* or a *Result Set Processing Table* is loaded in the *Function/Table Editor*.

The context sensitive menu for column headers includes the *Clear Selected* command along with:

Auto Column Width - automatically sets the column width to make all characters visible in all cells

Column Width - simply allows you to enter a column width, in pixels, for a single column.

The context sensitive menu for row headers also includes the *Clear Selected* command along with these other options:

Add Rows - adds the number of specified rows below the last row currently in the *Function/Table Editor*.

Insert Row - inserts a new row beneath the current row in the *Function/Table Editor*.

Delete Row- removes the current row from the *Function/Table Editor*.

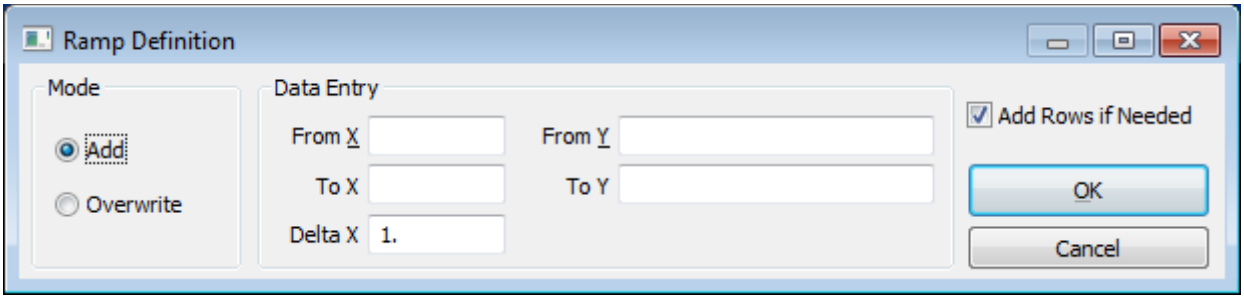
Row Height - allows you to enter a height, in pixels, for all rows in the *Function/Table Editor*.

Additional commands may be available on the column header and row header context-sensitive menus when a certain type of table is loaded in the *Function/Table Editor*.

Function-specific Context Sensitive Menu Commands

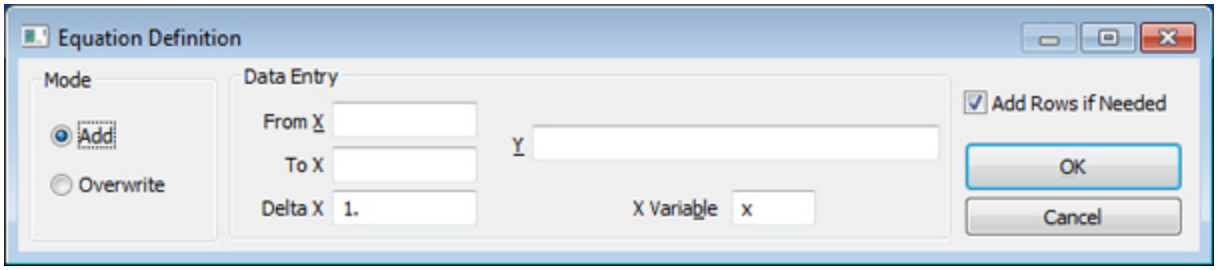
When a function is loaded in the *Function/Table Editor*, clicking the right mouse button will cause the context sensitive menu to appear, which contains commands designed to facilitate creation of functions.

Ramp... - displays the *Ramp Definition* dialog box, which allows you to define equally spaced points along a linear function:



When *Mode* is set to *Add*, values can be added to empty cells or new rows can be added to the end of an existing function. When *Mode* is set to *Overwrite*, cells in an existing function will be replaced with new values. Enter values for *X*, *Y*, *To X*, and *To Y*, along with the *Delta X* value. All data points between *X* and *To X* will be linearly interpolated at every multiple of *Delta X*. If *Add Rows if Needed* is enabled, which is the default, then rows will be automatically added to accommodate the entire linear ramp function. Otherwise, only the existing rows will be populated with values.

Equation... - displays the *Equation Definition* dialog box, which allows you to define more complex relationships than a simple linear ramp function.

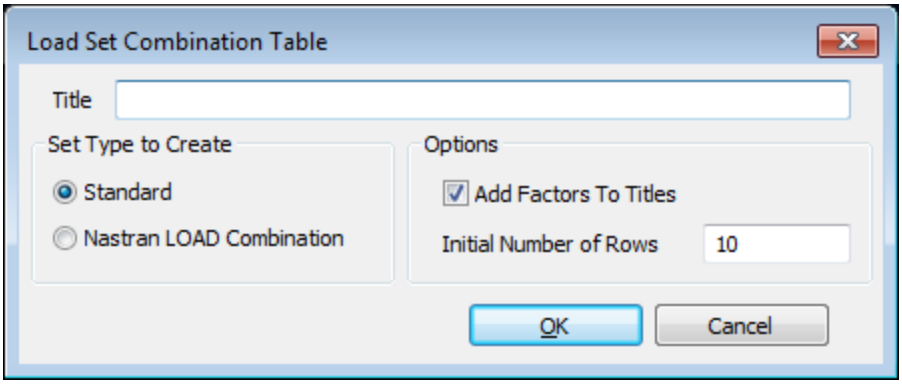


When *Mode* is set to *Add*, values can be added to empty cells or new rows can be added to the end of an existing function. When *Mode* is set to *Overwrite*, cells in an existing function will be replaced with new values. Similar to *Ramp...*, enter values for *X*, *To X*, and *Delta X*. However, for *Y*, type in any equation (in terms of the *X Variable* - !x by default) that defines the function that you want to represent. A typical example, might be setting *Y* to sin(!x). If you do not want to use the !x variable, it can be changed, but make sure that you use the new variable in your equations. If *Add Rows if Needed* is enabled, which is the default, then rows will be automatically added to accommodate the entire function. Otherwise, only the existing rows will be populated with values.

Chart Data Series... - displays the *Select Chart Data Series* dialog box, which allows you to select an existing Data Series, then uses the XY values of the selected Data Series to populate the cells in *Function/Table Editor*.

Load Set Combination Table...

When *Load Set Combination Table* is selected, the *Load Set Combination Table* dialog box will be displayed:



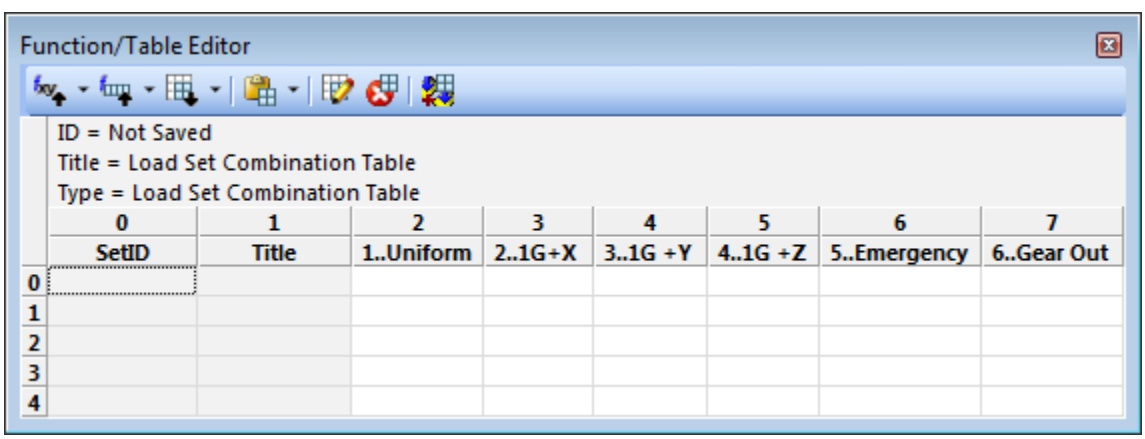
Use the *Set Type to Create* option to choose the type of load set(s) to create, *Standard* or *Nastran LOAD Combination* load sets. See Section 4.3.1, "Model, Load, Create/Manage Set..." for more information on *Load Set Type*. Combinations of existing *Nastran LOAD Combination* load sets can be created by using the referenced load sets of each *Nastran LOAD Combination* at the time of creation in the new *Nastran LOAD Combination* load set.

In the *Options* section, enable the *Add Factors To Titles* option to append the title of each new load set with ('scale factor'* 'existing load set ID', etc.) for each existing load set used by the new load set. If nothing is specified for *Title* in a particular row and this option is enabled, then the title will be ('scale factor'* 'existing load set ID', etc.). If no *Title* is given and this option is disabled, then the Load Set title will simply be 'Untitled'.

The *Initial Number of Rows* option is used to specify the number of rows, representing new load sets to create, which will initially appear in the *Function/Table Editor* after the existing load sets have been selected.

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

After clicking OK in the *Load Set Combination Table* dialog box, select the existing load sets to possibly use in combinations using the *Select Load Sets to Use in Combinations* dialog box, then click OK. The *Function/Table Editor* will now appear like this:



A *SetID* and *Title* may optionally be entered for each row. To include an existing load set in a new load set, simply enter a scale factor on a specific row in the appropriate column. Use 1.0 to simply include the existing load set with no scaling. Once all of the scale factors have been entered, click the *Table Combine Load Sets* icon in the *Function/Table Editor* or click the right-mouse button in any cell and choose the *Create Combined Sets* command from the context-sensitive menu to create the new load sets

.

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

Context Sensitive menus for Load Set Combination Table

Additional commands on the context-sensitive menu for cells:

Create Combined Sets - used to create the new load sets. All of the load set combinations currently defined in the *Function/Table Editor* will be created at the same time.

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

Activate - makes the load set represented by the column the active load set in the model.

List Load Set - lists information about the load set represented by the column including individual loads in the load set.

Sum Forces in Load Set - lists the load summation for the load set represented by the column using the same format as the *Tools, Check, Sum Forces* command. See Section 7.4.5.10, “Tools, Check, Sum Forces...”.

Add Load Sets - adds load sets selected via the *Select Load Sets to Use in Combinations* dialog box as new columns in the *Function/Table Editor.*

Change Load Set - updates load set referenced by the current column in the *Function/Table Editor.*

Remove Selected Load Sets - removes load sets corresponding to the columns currently selected in the *Function/Table Editor.*

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

List Combination - lists information about the potential load set combination represented by the row including individual loads in the various load sets.

Sum Forces in Combination - lists the load summation for the potential load combination represented by the row using the same format as the *Tools, Check, Sum Forces* command. See Section 7.4.5.10, “Tools, Check, Sum Forces...”.

Example

This type of table can be used to create 5 new load sets from 6 existing load sets. Simply enter scale factor values into various cells (optionally, a specific *SetID* and *Title* can be entered for each row):

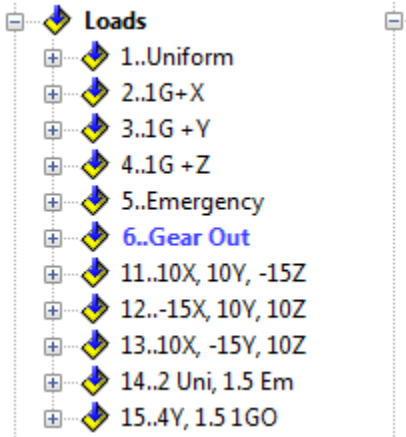
Function/Table Editor

ID = Not Saved
Title = Load Set Combination Table
Type = Load Set Combination Table

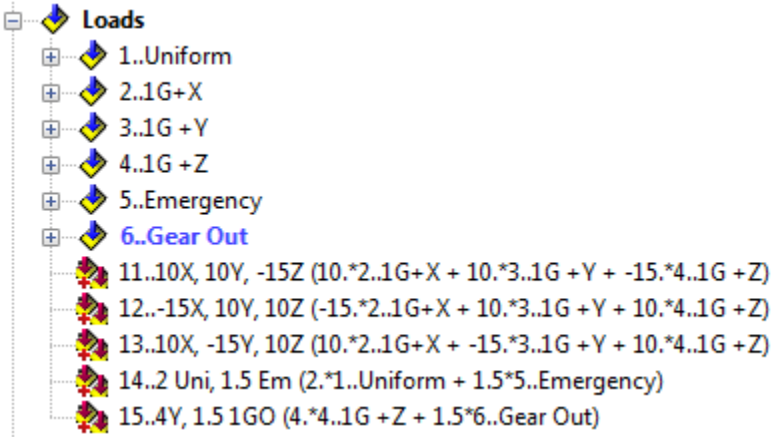
	0	1	2	3	4	5	6	7
	SetID	Title	1..Uniform	2..1G+X	3..1G +Y	4..1G +Z	5..Emergency	6..Gear Out
0	11	10X, 10Y, -15Z		10.	10.	-15.		
1	12	-15X, 10Y, 10Z		-15.	10.	10.		
2	13	10X, -15Y, 10Z		10.	-15.	10.		
3	14	2 Uni, 1.5 Em	2.				1.5	
4	15	4Y, 1.5 1GO				4.		1.5

Once all the scale factors have been entered, select any cell in the *Function/Table Editor*, then click the *Create Combined Load Sets* icon or right-mouse click and choose *Create Combined Sets*. The following Load Sets will be created from the table shown above:

Titles Defined
Add Factors To Titles = Off
Set Type to Create = Standard



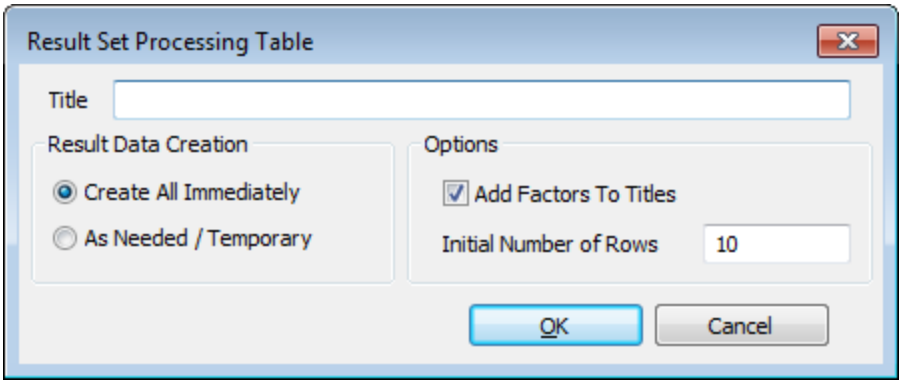
Titles Defined
Add Factors To Titles = On
Set Type to Create = Nastran LOAD Combination



Result Set Processing Table...

Any combination of the “Linear Combination”, “RSS Combination”, and/or the various “Envelope” methods may be used in a single table to create or update previously created output sets. See Section 8.5.6, “Model, Output, Process” for more information on these methods.

Settings for the *Results Set Processing Table* are specified using the *Results Set Processing Table* dialog box:



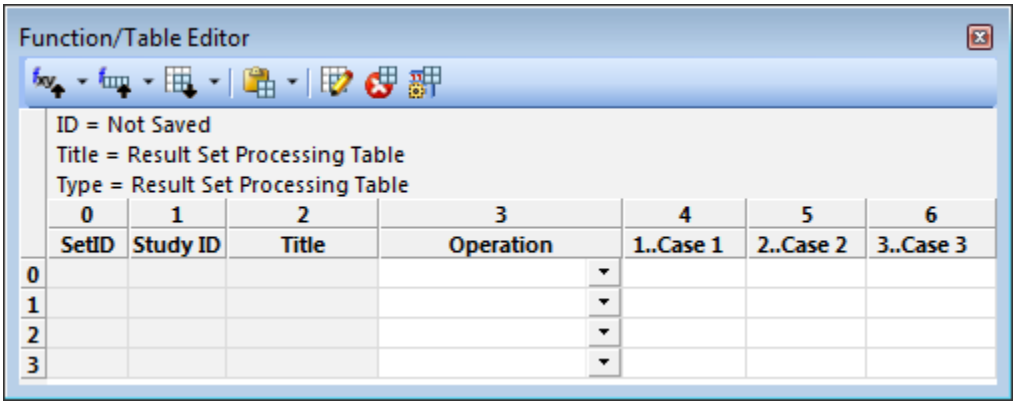
Use the *Results Data Creation* option to choose the method for creating the output set(s), *Create All Immediately* or *As Needed/Temporary.* When using *Create All Immediately* the output values will be stored in the created or updated output set(s). Output sets created with this option will be displayed 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, enable *Add Factors To Titles* 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 enabled, then the title will be “‘Type of Operation’ (‘scale factor’* ‘existing output set ID’, etc.)”. If no *Title* is given and this option is disabled, 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 *Function/Table 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 *Results Set Processing Table* 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 *Function/Table 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? Table 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 *Function/Table Editor*, then a question will be asked, "Ok to Combine? Combining will delete Result Sets used in this Table 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 processed 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 Table

Additional commands on the context-sensitive menu for cells:

Apply to Study - if a table 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 table. When successful, the output set columns currently in the *Function/Table 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 *Function/Table Editor* will be done at the same time.

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

Activate - makes the output set represented by the column the active output set in the model.

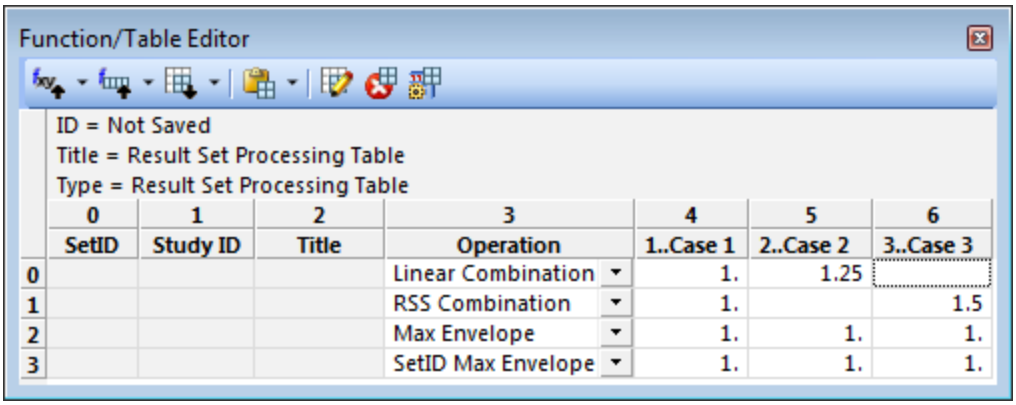
Add Result Sets - allows you to add columns by selecting any number of output sets using a dialog box

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

Remove Selected Result Sets - removes output sets corresponding to the columns currently selected in the *Function/Table Editor.*

Example

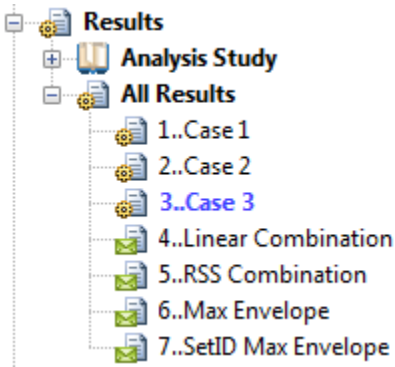
This type of table can be used to create 4 new output sets, each using a different Operation (a *Linear Combination*, a *RSS Combination*, a *Max Envelope*, and a *SetID Max Envelope*), using 3 existing output sets. Simply enter scale factor values into various cells. It is typical to simply use 1.0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



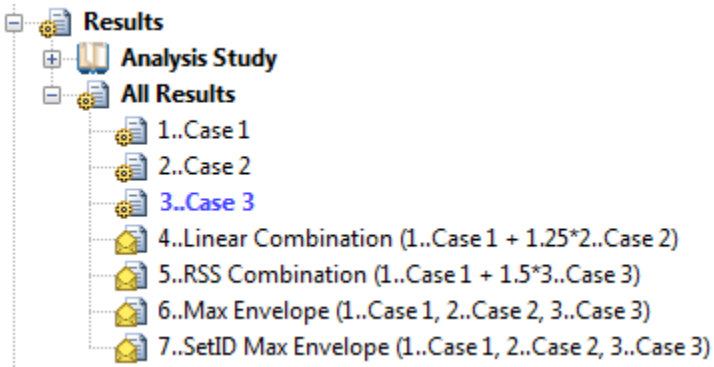
	0	1	2	3	4	5	6
	SetID	Study ID	Title	Operation	1..Case 1	2..Case 2	3..Case 3
0				Linear Combination	1.	1.25	
1				RSS Combination	1.		1.5
2				Max Envelope	1.	1.	1.
3				SetID Max Envelope	1.	1.	1.

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 *Function/Table 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.

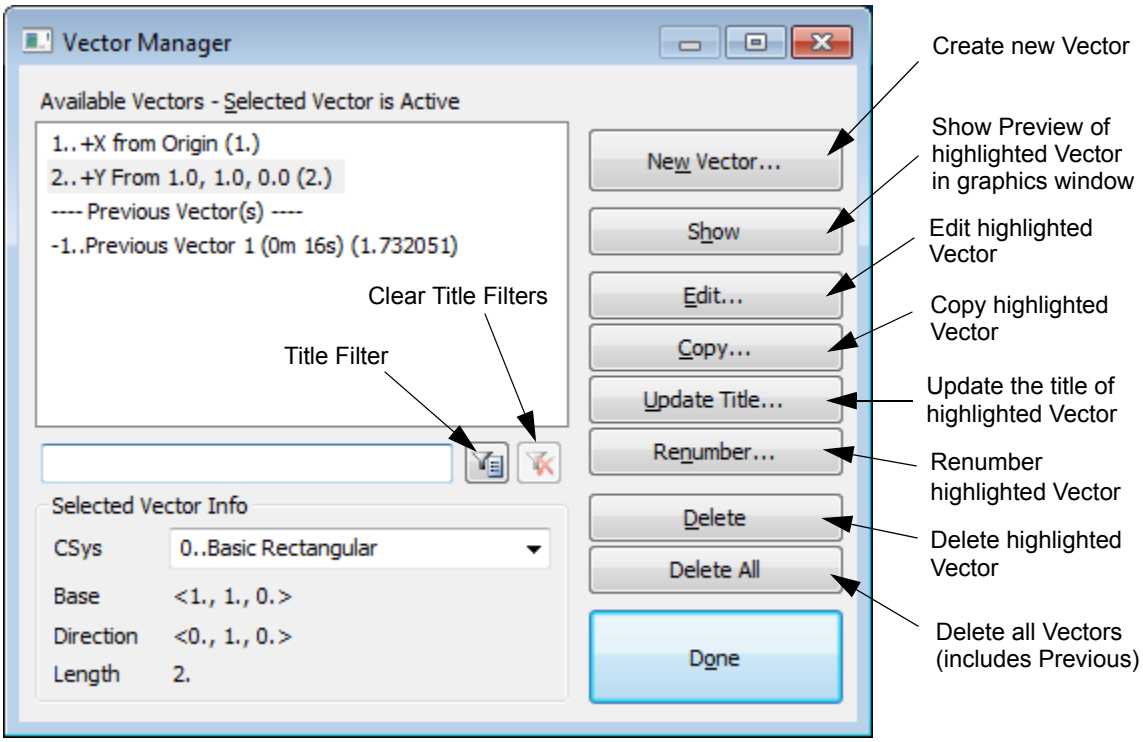
- Added *Tools, Vector Manager...* command.

... displays the *Vector Manager* dialog box which may be used to create a new vector, show a preview of a selected vector in the graphics window, edit all attributes of an existing vector, copy an existing vector, update the *Title* of an existing vector, renumber an existing vector, delete a single vector, or delete all vectors. Once a vector has been saved, it can then be used during a command via the *Saved* option on the *Method^* menu in the standard vector definition dialog box.

The *Available Vectors...* list contains all vectors which have been created using the *Vector Manager* at the top of the list, along the *Previous Vector(s)* section, which contains the last 5 vectors which were defined during commands.

Once 5 vectors exist in the *Previous Vector(s)* section, the next vector defined during a command will appear at the top of the *Previous Vector(s)* section, with an ID of -1, the other vectors in this section will move down by one, and the vector which previously had an ID of -5 will be removed from the section. Also, the length of each vector will appear in parentheses after the title of each vector in the list.

The *Selected Vector Info* section lists the *Base* location and *Direction* of the vector currently highlighted in the *Available Vectors...* list, relative to the coordinate system specified in the *CSys* drop-down, along with the *Length*.



Clicking *New Vector* will open the *New Vector* dialog box, which contains *ID* and *Title* fields, along with the *Unit Vector* option, which when enabled, insures the vector being specified will always have a length of 1.0. Clicking OK will then display the standard vector definition dialog box and a vector may be created using any of the available methods found on the *Methods^* menu.

Once a "saved" vector is highlighted in the *Available Vectors...* list: *Edit* offers the ability to redefine all of the attributes of the vector, other than ID; *Copy* simply creates a copy of the vector; *Update Title* displays a dialog box which can be used to update the title of the vector; *Renumber* displays a dialog box to enter a new ID (if a vector using the supplied ID already exists, the next available ID will be used); and *Delete* deletes the vector from the model. If instead, a vector is highlighted in the *Previous Vector(s)* section, *Save* (typically the *Edit* button) displays the *New Vector* dialog box, with all fields and options available for modification, before the vector is saved.

Finally, *Delete All* will delete all of the "saved" vectors and any vectors currently in the *Previous Vector(s)* section.

Filter/Clear Filter buttons

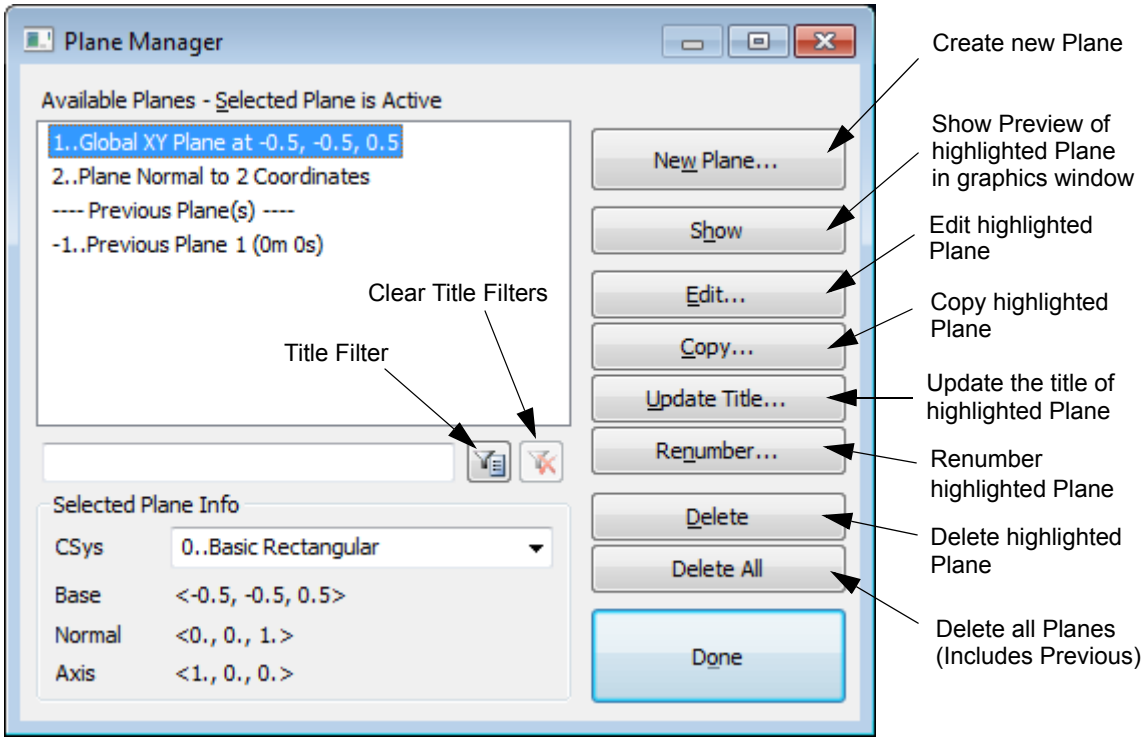
To reduce the number of vectors being displayed in the *Available Vectors...* list, enter text into the field below the list, then click the *Title Filter* button and only those vectors that contain the specified text will remain in the list. If additional text is entered and the *Filter* icon button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

- Added *Tools, Plane Manager...* command.

... displays the *Plane Manager* dialog box which may be used to create a new plane, show a preview of a selected plane in the graphics window, edit all attributes of an existing plane, copy an existing plane, update the *Title* of an existing plane, renumber an existing plane, delete a single plane, or delete all planes. Once a plane has been saved, it can then be used during a command via the *Saved* option on the *Method^* menu in the standard plane definition dialog box.

The *Available Planes...* list, starting at the top, contains all planes which have been created using the *Plane Manager*, along with the *Previous Plane(s)* section, which contains the last 5 planes which were defined during commands. Once 5 planes exist in the *Previous Plane(s)* section, the next plane defined during a command will appear at the top of the *Previous Plane(s)* section, with an ID of -1, the other planes in this section will move down by one, and the plane which previously had an ID of -5 will be removed from the section.

The *Selected Plane Info* section lists the *Base* location along with the *Normal* and *Axis* vectors of the plane currently highlighted in the *Available Planes...* list, relative to the coordinate system specified in the *CSys* drop-down.



Clicking *New Plane* will open the *New Plane* dialog box, which contains *ID* and *Title* fields. Clicking OK will then display the standard plane definition dialog and a plane may be created using any of the available methods found on the *Methods^* menu.

Once a "saved" plane is highlighted in the *Available Planes...* list: *Edit* offers the ability to redefine all of the attributes of the plane, other than ID; *Copy* simply creates a copy of the plane; *Update Title* displays a dialog box which can be used to update the title of the plane; *Renumber* displays a dialog box to enter a new ID (if a plane using the supplied ID already exists, the next available ID will be used); and *Delete* deletes the plane from the model. If instead, a plane is highlighted in the *Previous Plane(s)* section, *Save* (typically the *Edit* button) displays the *New Plane* dialog box, with all fields available for modification, before the plane is saved.

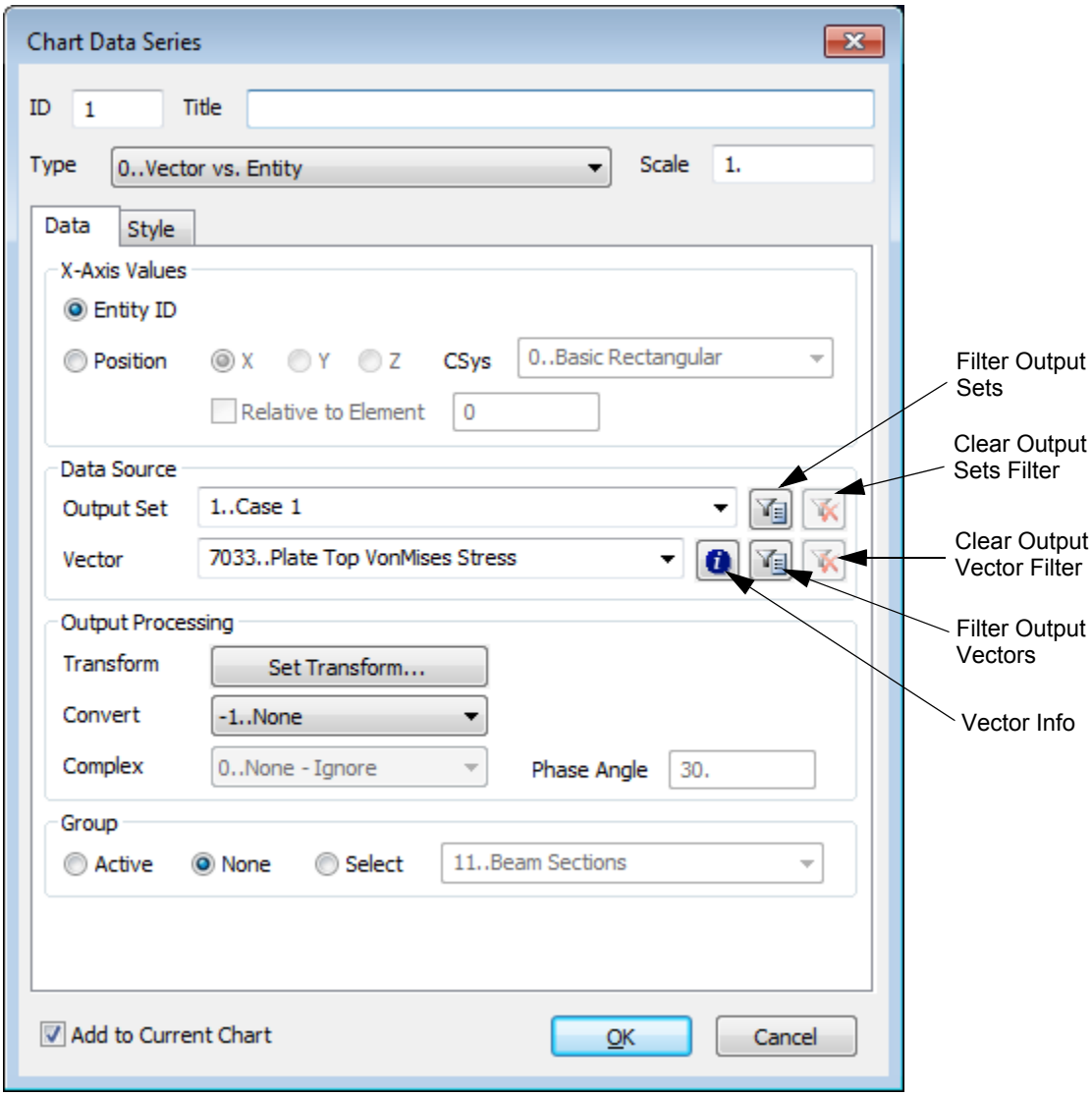
Finally, *Delete All* will delete all of the "saved" planes and any planes currently in the *Previous Plane(s)* section.

Filter/Clear Filter buttons

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

- Updated *Tools, Check, Element Quality* command by adding *Tetra EPIA*, *Pyr EPIA*, *Penta EPIA*, and *Hex EPIA* to the *NX Nastran* tab of the *Check Element Quality* dialog box.
- Added *Y Complex* radio button to the *Chart Axes* tab of the *Charting* dialog box. When selected, all of the options are shown for *Y Complex*, but only a subset can be modified, as the others are inherited from *Y Axis*.
- Updated *Chart Data Series* dialog box.

One way to create a new *Data Series* is to simply select *New Data Series* in the *Chart Data Series Manager*. An additional method is also available via the *Add Data Series* command found on both the *Chart Options* icon menu and the context-sensitive menu for the *Chart* itself. Both methods access the *Chart Data Series* dialog box



Like many other entities in FEMAP, each *Data Series* must have a unique *ID* and may optionally have a unique *Title*, which may be entered at the top. Also, a *Scale* value can be entered to "scale" only the "Y" values. Use the *Type* drop-down to select the "Type" of *Data Series* to create. The *Data* tab for each type of *Data Series* is unique and allows you to select and specify appropriate data. In contrast, the *Style* tab is the same for all types. Both tabs will be discussed in detail below. Finally, when the *Add to Current Chart* toggle at the bottom of the dialog box is enabled, any newly created *Data Series* will automatically be displayed on the *Chart* currently loaded in the *Charting* pane, once *OK* has been pressed.

Data Tab (Each Data Series Type is Different)

This section is used to select the "Type" of *Data Series* by selecting one of the five options from the *Type* drop-down. Depending on the type selected, different options will be available. The five types are "0..Vector vs Entity", "1..Vector vs. Output Set", "2..Vector vs. Vector", "3..Expand Complex", and "4..Function".

Output Filters and Vector Info

All of the types except *Function* include *Filter Output Sets* and *Filter Output Vectors* icon buttons which can be used to reduce the amount of output data available for selection, while the *Vector Info* icon button can be used to bring up the *Output Set/Vector Info* dialog box. See Making it Easier to Select Output Data - Using the Output Vector Filters and Vector Info for more information.

Output Processing

All of the types except *Function* contain an *Output Processing* section. This section allows you to:

Transform - click the *Set Transform* button to transform nodal or elemental output values using the same on-the-fly transformation options available when creating a contour plot. For more information, see Transforming Your Output and Contour Output Vector Transformation.

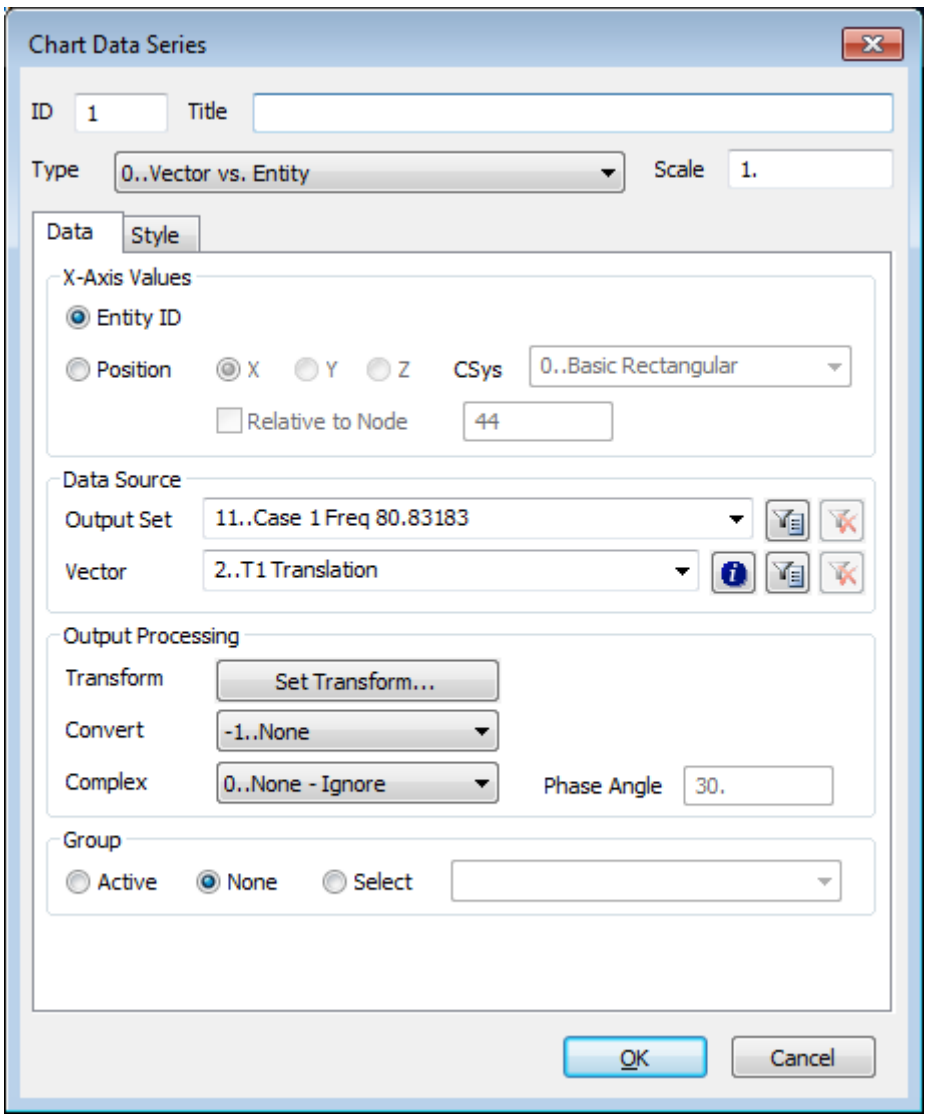
Convert - convert nodal output to elemental output, and vice versa. This option uses the same process as the *Convert* tab of the *Model, Output, Process* command, with "0..Average" and "1..Max Value" corresponding to the options in the *Conversion Approach* section, but does it on-the-fly instead of creating new output vectors. For more information, see Section 8.5.6.7, "Converting Nodal and Elemental Data".

Complex - enables you to convert complex data to real data, on-the-fly, when post-processing complex results, which can be set using the *Complex Results...* button in the Select PostProcessing dialog box or via the PostProcessing Toolbox. For more information, see Complex Results...

By default, this option is set to "0..None Ignore". The "1..Match Model" option corresponds to using the *Use Model Options* option in the *Model Settings* section of the *Complex Results Options* dialog box, while "2..Match View" corresponds to using *Synchronize Phase* in the *View Override* section. Finally, "3..Match Phase Angle" uses the value set for *Phase Angle* on the *Data* tab of the *Chart Data Series* dialog box.

Vector vs. Entity

Plots XY data as a function of *ID* number or position of nodes or elements in an axis direction for a single *Output Vector* in one *Output Set*.



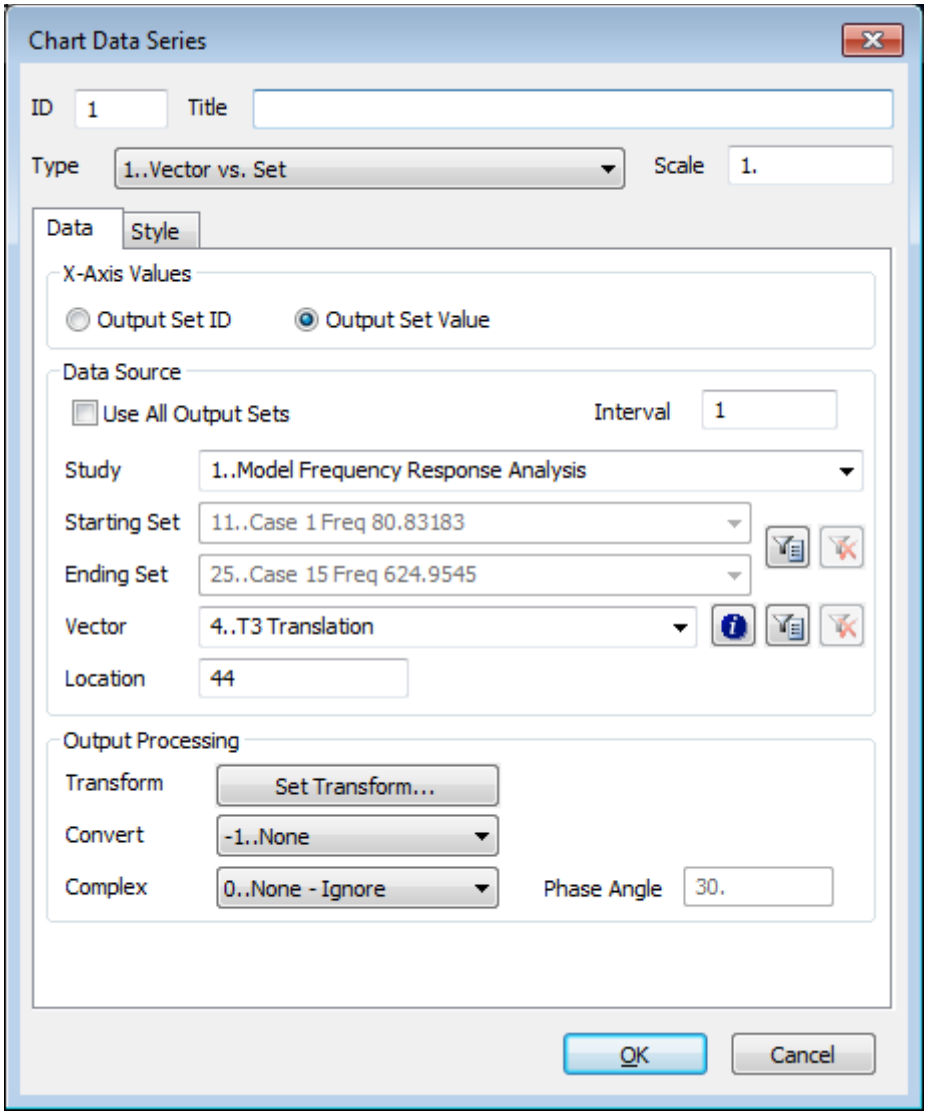
The *X-Axis Values* section offers two options, *Entity ID* or *Position*. When *Entity ID* is selected, the node or element ID of each data point will be used as the X value. When *Position* is selected, the *X*, *Y*, or *Z* coordinate of each node or the centroid of each element will be used as the data point X value. The *CSys* drop-down can be used to use the *X*, *Y*, or *Z* coordinates of an entity in a selected coordinate system. Use the *Relative to Node* option to have the values be calculated relative to a specified node when looking at a nodal output vector (*Relative to Element* works similar for an elemental vector).

The *Data Source* section is simply used to select an *Output Set* and an output *Vector* for the *Data Series*, while the *Output Processing* section is described above in Output Processing.

The *Group* section is used to limit the plot of a *Data Series* to a certain portion of the model. Selecting *None* will display data for the entire model. *Active* will display data for the nodes or elements in the "active" group, or for the entire model if no group is active. If *Select* is chosen, a group must also be selected from the drop-down list.

Vector vs. Output Set

Plots XY data versus the *Output Set ID* or *Output Set Value* for a single *Output Vector* across several *Output Sets*.



The *X-Axis Values* section offers two options, *Output Set ID* or *Output Set Value*. When *Output Set ID* is selected, the IDs of the specified output sets will be used as the X values, while *Output Set Values* will use the values of the specified output sets as the X values.

The *Data Source* section is used to specify a range of output sets, an output vector, and a *Location* for the *Data Series*. By default, the *Use All Output Sets* option will be "on", which will simply use all available output sets in the model. If *Use All Output Sets* option is "off", a range of output sets can be specified by selecting an Analysis Study

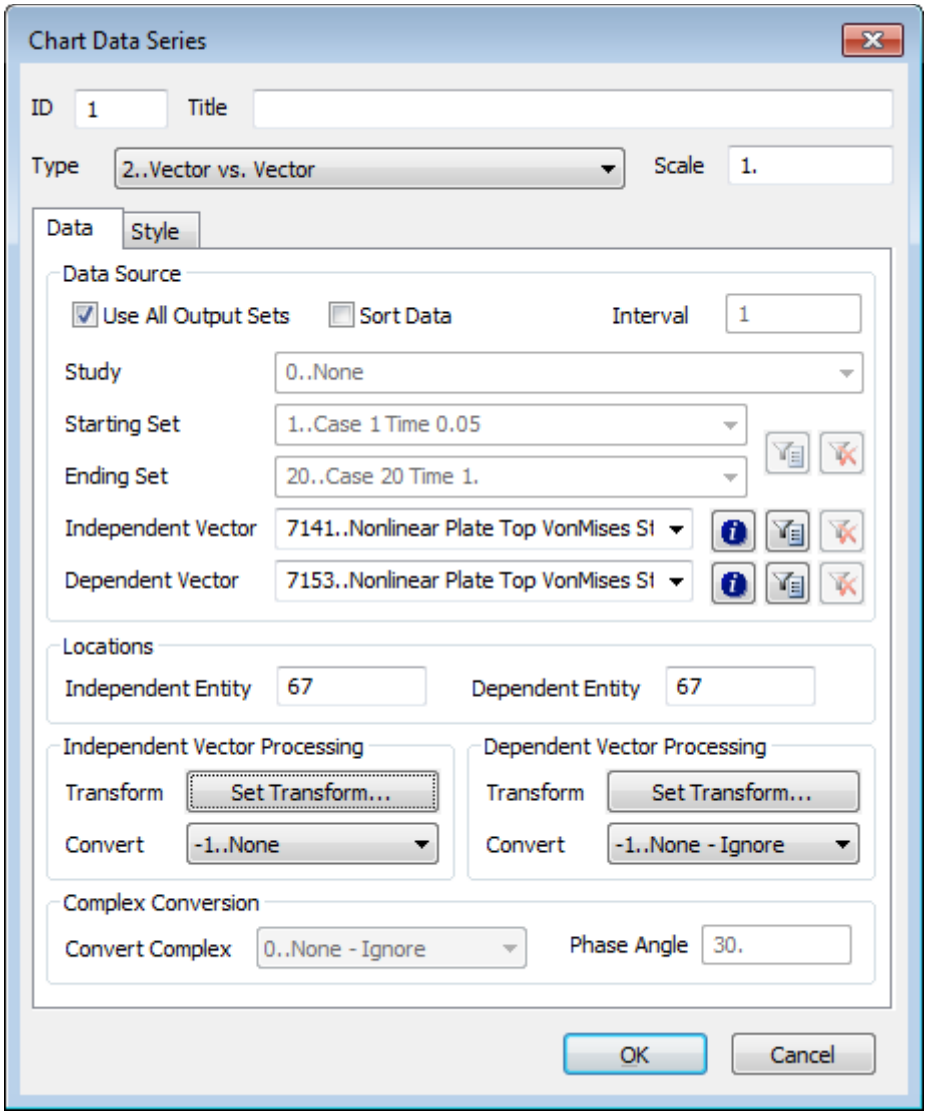
from the *Study* drop-down or by using the *Starting Set* and *Ending Set* drop-downs. Optionally, an *Interval* value can be specified, so only every n-th output set in the range will be used.

The *Vector* drop-down is used to select the output vector of interest, while *Location* must be the ID of a node or element in the model. For *Location*, a node ID must be specified when a nodal output vector is selected, while an element ID must be specified for an elemental output vector. The *Select Multiple* button can be used to create any number of new *Data Series* which all share the same *Output Set* and *Output Vector* data, but a different *Location* value based on selected node or element IDs.

The *Output Processing* section is described above in Output Processing.

Vector vs. Vector

Plots XY data for output from an *Independent Vector* at a specified entity versus output from a *Dependent Vector* at a specified entity across any number of *Output Sets*.



The *Data Source* section is used to specify a range of output sets, an *Independent Vector* (X-axis output vector), and a *Dependent Vector* (Y-Axis output vector) for the *Data Series*. By default, the *Use All Output Sets* option will be enabled, which will simply use all available output sets in the model. If *Use All Output Sets* option is disabled, a range of output sets can be specified by selecting an Analysis Study from the *Study* drop-down or by using the *Starting Set* and *Ending Set* drop-downs. Optionally, an *Interval* value can be specified, so only every n-th output

set in the range will be used. Also, The *Sort Data* option can be used to display X-values in ascending order (lowest to highest), regardless of how the data exists in the range of output sets.

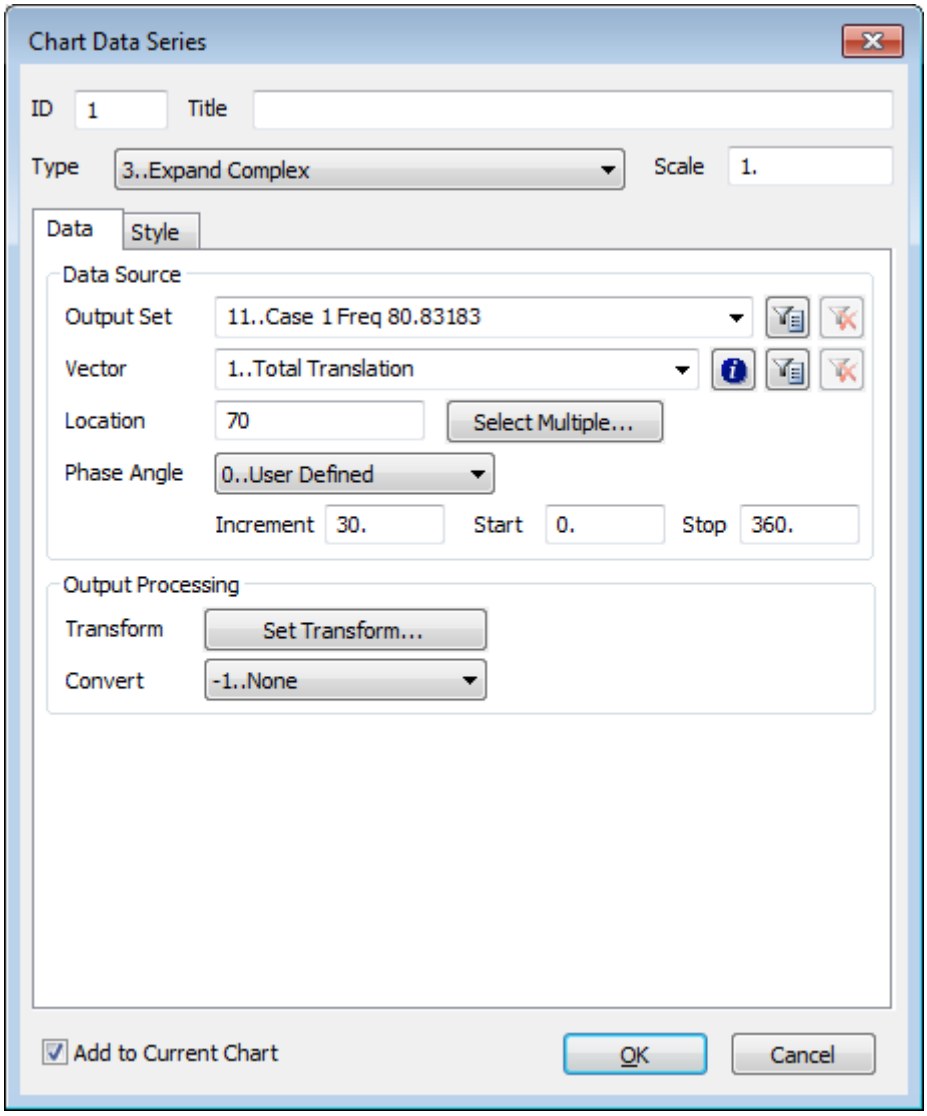
Note: When *Sort Data* is on, lines will be drawn through the data points based on ascending X-values. Also, when listing values or sending them to the clipboard, the X-values will be in ascending order.

The *Locations* section is used to specify an Independent Entity and a Dependent Entity, which must be a node or element ID in the model. A node ID must be specified when a nodal output vector is selected, while an element ID must be specified for an elemental output vector. The *Independent Entity* and the *Dependent Entity* may be the same entity.

The settings for the *Transform* and *Convert* options in the *Independent Vector Processing* and *Dependent Vector Processing* sections, which can be different from one another, along with the *Convert Complex* option in the *Complex Conversion* section, are described above in Output Processing.

Expand Complex

Plots XY data of complex results which have been expanded across a phase range to allow you to view the results at various phase intervals, instead of a single component of magnitude/phase:



The *Data Source* section is used to select an *Output Set*, an output *Vector*, and a *Location* for the *Data Series*. The *Vector* drop-down is used to select the output vector of interest, while *Location* must be the ID of a node or element in the model. For *Location*, a node ID must be specified when a nodal output vector is selected, while an element ID must be specified for an elemental output vector. The *Select Multiple* button can be used to create any number of

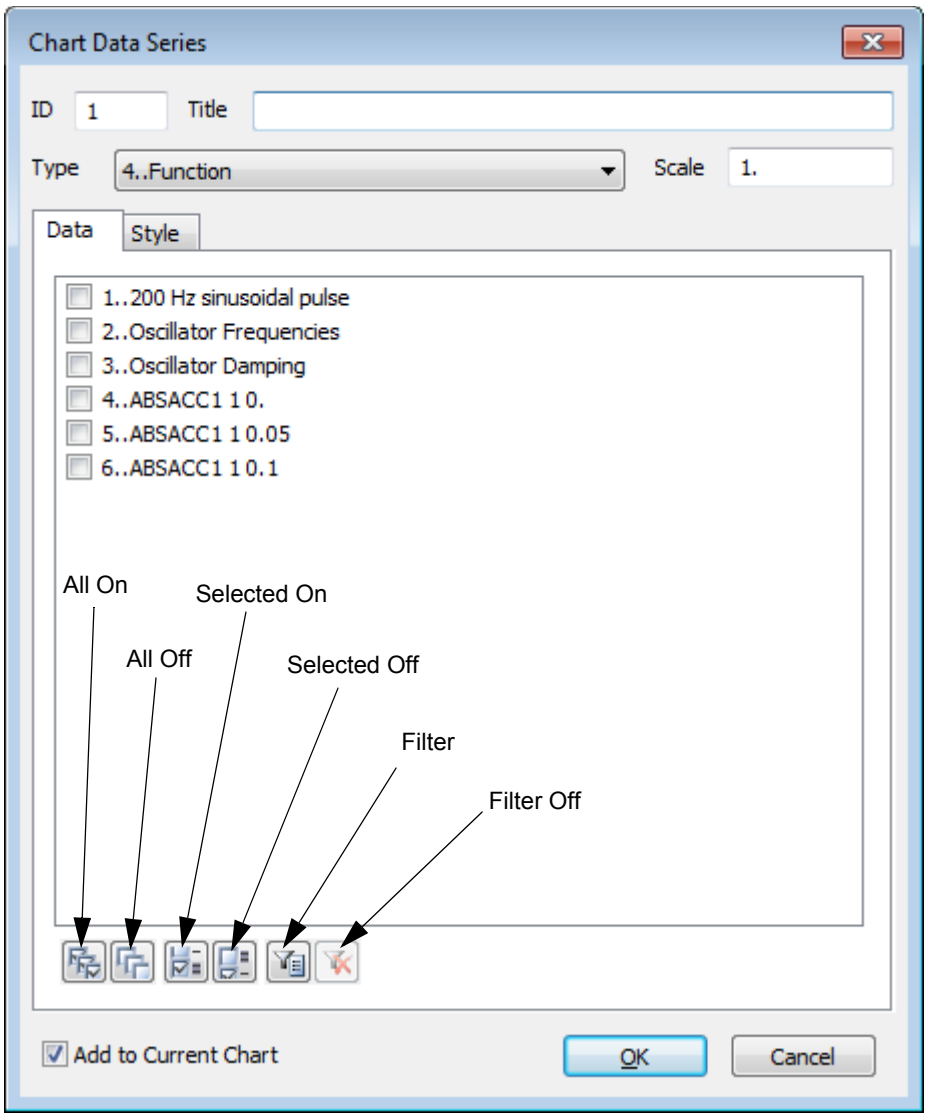
new *Data Series* which use all of the current settings, except for *Location*. The *Location* for each *Data Series* is determined based on selected node or element IDs.

The setting in the *Phase Angle* drop-down specifies how the phase range used for the expansion of complex results is determined. When set to “0..User Defined”, the values for *Increment*, *Start*, and *Stop* in the *Data Source* section will be used to determine the phase range. The “1..Match Model” option corresponds to using the to using the *Use Model Options* option in the *Model Settings* section of the *Complex Results Options* dialog box, while “2..Match View” corresponds to using *Synchronize Phase* in the *View Override* section. For more information, see Complex Results...

The *Transform* and *Convert* options in the *Output Processing* section are described above in Output Processing.

Function

Plots XY data for a *Function*. Not always a Post-Processing option. To create multiple *Data Series* based on Functions at once, simply check the desired functions in the list:

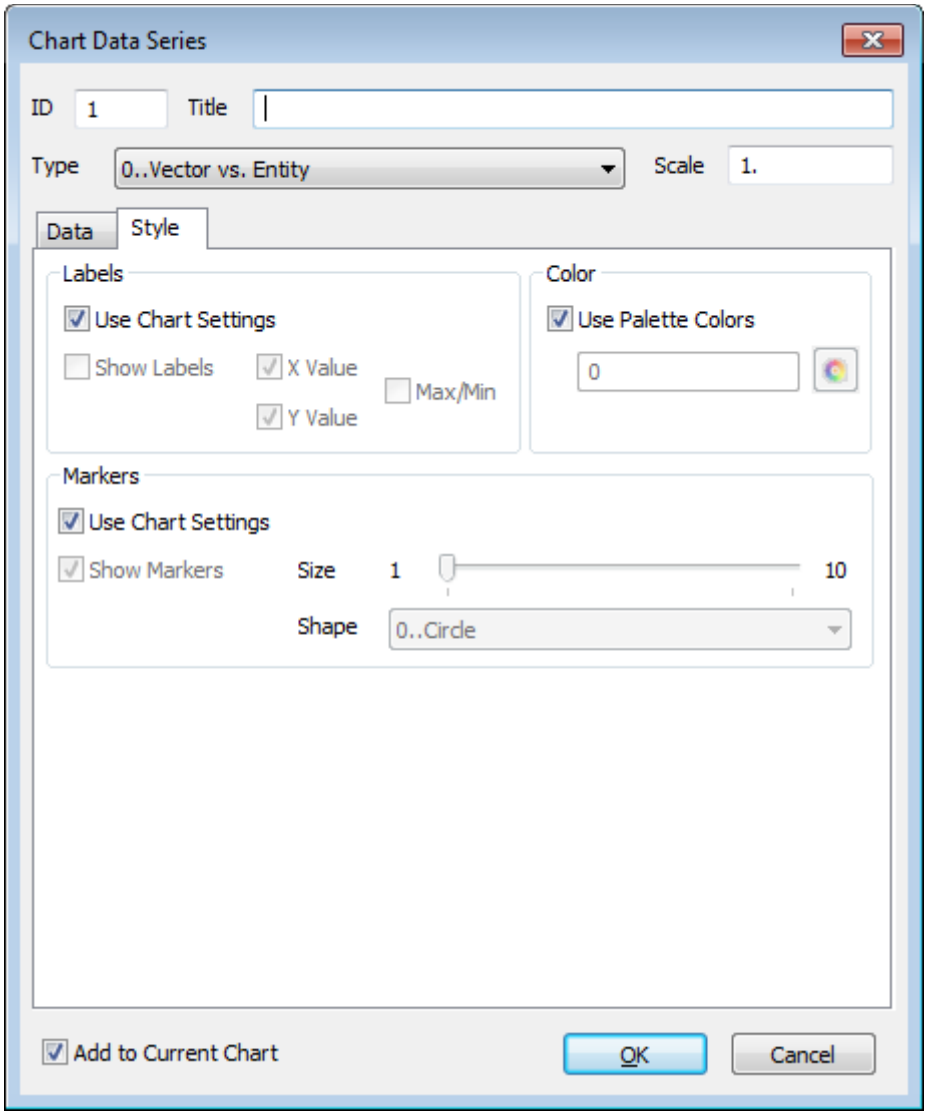


It is also possible to highlight any number of Functions from the list. Holding down the “Ctrl” key will enable highlighting of multiple Functions. Holding down “Shift” while picking a “first” Function, then a “last” Function will highlight multiple. Once highlighting is complete, use the *Toggle Selected On* or *Toggle Selected Off* icon buttons to check or uncheck the highlighted Functions. Once multiple Functions are selected, click OK to create an individual *Data Series* for each selected function. In addition, the *All On* icon button may be used to select all of the Functions, while *All Off* will set the check boxes to foe all Functions to off.

To reduce the number of Functions being displayed in the list, enter text into the field below the list, then click the *Filter* icon button and only those Functions that contain the specified text will remain in the list. If additional text is entered and the *Filter* icon button clicked again, the number of Functions in the list may be further reduced. To display all available Functions in the list again, click the *Filter Off* icon button.

Style Tab (all Data Series Types)

This tab is used to set options for *Labels*, *Color*, and *Markers* for each individual *Data Series*.



By default, the *Use Chart Settings* option is enabled for all three sections of the *Style* tab, so the options in each subsection only become available when *Use Chart Settings* is disabled. In addition, if the *Use Chart Settings* or *Use Palette Colors* options are on in the *Charting* dialog box, then the *Title* of each subsection will be appended with (*Chart Label Settings/Marker Settings/Palette Used*).

Note: Alternatively, the *Use Chart Settings - All* option may be set on both the *Data Series Markers* or *Data Series Labels* icon menu, while *Use Palette Colors - All* may be set via the *Chart Colors* icon menu.

Labels - This section controls the display and appearance of *Labels* for the *Data Series*. *Labels* are always the same color as the associated *Data Series*. The options in this dialog box only affect *Labels* for a single *Data Series*.

Use Chart Settings - When "on", all *Labels* options are controlled by the *Chart*. When "off" the settings are used only for the *Data Series* currently being created or edited.

Show Labels - on/off toggle controls display of *Labels*.

Show X Value - on/off toggle for *X Value* in the displayed label.

Show Y Value - on/off toggle for *Y Value* in the displayed label.

Max/Min Only - when “on”, only displays labels at the Maximum and Minimum Y values, otherwise all data points are labeled.

Color - This section controls the *Color* and optionally *Line Style* of an individual *Data Series*.

Use Palette Colors - When “on”, all *Color* options are controlled by *Palette* in the *Chart*. When “off” the *Color* is used only for the *Data Series* currently being created or edited.

Palette button - Used to select the *Color* and optionally a *Line Style* from the *Color Palette*.

Markers - This section controls the display and appearance of *Markers* for the *Data Series. Markers* are always the same color as the associated *Data Series*. The options in this dialog box only affect *Markers* for a single *Data Series*.

Use Chart Settings - When “on”, all *Markers* options are controlled by the *Chart*. When “off” the settings are used only for the *Data Series* currently being created or edited.

Show Markers - on/off toggle controls display of *Markers*.

Size - slider bar controls size of *Markers*, from *1- 10*.

Shape - controls shape of *Markers*, choose from *Circle*, *Square*, *Diamond*, *Triangle*, *Pentagon*, *Hexagon*, or *Star.*

- Added *Complex Plot* drop-down to the *Chart Settings* tab of the *Charting* dialog box, which offers the ability to display both magnitude and phase data or real and imaginary data in a single *Chart,* by adding a secondary plot which uses the same X Axis as the primary plot. By default, this option is set to “0..Off”. The other settings are used to select the position of secondary plot, which can appear above (“1..Top”) or below (“2..Bottom”) the primary plot. When plotting magnitude and phase data in the same chart, phase data is always displayed on the secondary plot. When plotting real and imaginary data in same chart, imaginary data is always displayed on the secondary plot.

Note: When a secondary plot is visible, “type of data” Titles are always displayed for the *Y Axis* and *Y Complex* axis. If the *Y Axis* has a defined *Title*, then “type of data” Titles are also shown, in parentheses.

OLE/COM API

New and modified API Objects and Attributes

- Added Plane (fePlane) object to the API. Also, added title, base, vBase, norm, vNorm, axis, and vAxis attributes to the Plane Object.
- Added Vector (feVector) object to the API. Also, added title, base, vBase, dir, vDir, and Length attributes to the Vector Object.
- Added Table Data (feTableData) object to the API. Also, added Title, Type, Subtype, FunctionType, VectorFunctionType, Rows, and Columns to the Table Data Object.
- Added User Defined Graphics (feUserDefinedGraphics) object to the API.
- Added Pt1 and Pt4 attributes to the feAeroPanel Object.
- Added NasCurvatureStrain attribute to the feAnalysisCase Object.
- Added NasCurvatureStrain attribute to the feAnalysisMgr Object.
- Added ComplexPlotLocation, AxisAutoscale2, vAxisAutoscale2, AxisRange2, vAxisRange2, AxisRangePad2, vAxisRangePad2, AxisStyle2, vAxisStyle2, AxisLabelFormat2, vAxisLabelFormat2, AxisLabelDecimal2, and vAxisLabelDecimal2 attributes to the feChart Object.
- Added ConvertMethod, ConvertMethod2, TransformNodalMode, TransformNodalMode2, TransformNodalCSys2, TransformNodalCSys, TransformPlateMode, TransformPlateCSys, TransformPlateDOF, TransformPlateVector, vTransformPlateVector, TransformPlateTolerance, TransformPlateMode2, TransformPlateCSys2,TransformPlateDOF2, TransformPlateVector2, vTransformPlateVector2, TransformPlateTolerance2, ComplexMethod, ComplexPhase, ComplexStart, and ComplexEnd attributes to the feChartSeries Object.

- Added DataConversion attribute to the feDataSurf Object.
- Added DrawModeSelectFromAll attribute to the feDrawErase Object.
- Added NastranTetraEPIAOn, NastranTetraEPIALimit, NastranHexEPIAOn, NastranHexEPIALimit, NastranPenEPIAOn, NastranPenEPIALimit, NastranPyrEPIAOn, and NastranPyrEPIALimit attributes to the feElementQuality Object.
- Added DataConversion attribute to the feMapOutput Object.
- Added b and vb attributes to the feTMGBC Object.
- Added SecondaryRotationAxesOption attribute to the feViewOrient Object.

New and Updated API Methods

- Added SelectIDWithNew as a Common Entity method.
- Added GetEntitySet to the feConnectionRegion object.
- Added ElementHasThickness to the feElem object.
- Added GetNastranTetraEPIA, NastranTetraEPIA, GetNastranHexEPIA, NastranHexEPIA, GetNastranPenEPIA, NastranPenEPIA, GetNastranPyrEPIA, and NastranPyrEPIA to the feElementQuality object.
- Added Show to the feFreebody object.
- Added ShowAutoscale and SelectIDWithNew to the feSet object. In addition, added new “Saved Set” method, including SetDeveloperID, GetSavedSet, GetAllSavedSets, PutSavedSet, DeleteSavedSet, DeleteAllSavedSets, DeleteOtherSavedSets, ResetSavedSet, NextSavedSet, EmptySavedSet, CountSavedSets, AddToSavedSet, AddSetToSavedSet, RemoveFromSavedSet, and RemoveSetFromSavedSet.
- Added PutAll to the fePlane object.
- Added GetFunction, PutFunction, CellRange, SaveToFile, Initialize, Resize, Clear, SetCellDouble, SetCellInteger, SetCellText, SetCellEquation, SetCellEntity, GetCellDouble, GetCellInteger, GetCellText, GetCellEntity, SetMultiCellDouble, SetMultiCellInteger, SetMultiCellText, GetMultiCellDouble, and GetMultiCellInteger to the feTableData object.
- Added GetAll2 and PutAll2 to the feTMGBC Object.
- Added GetAll2 and PutAll2 to the feTMGCtrl Object.
- Added SetView, GetView, SetName, Publish, Blank, CollectorPoints, CollectorBitmap, CollectorSymbol, CollectorSymbolREAL8, CollectorText, CollectorTextINT4, CollectorTextREAL8, CollectorLines, CollectorTriangles, CollectorBlank, CollectorLabels, CollectorMarkForDeletion, CollectorAppearance, CollectorAddPointLocations, CollectorAddPointEntityLocations, CollectorAddPointEntityFaceLocations, CollectorAddSymbolLocations, CollectorAddSymbolEntityLocations, CollectorAddSymbolEntityFaceLocations, CollectorAddSymbolEntityFaceNormalLocations, CollectorAddSymbolREAL8Locations, CollectorAddSymbolREAL8EntityLocations, CollectorAddSymbolREAL8EntityFaceLocations, CollectorAddSymbolREAL8EntityFaceNormalLocations, CollectorAddTextLocations, CollectorAddTextEntityLocations, CollectorAddTextEntityFaceLocations, CollectorAddTextINT4Locations, CollectorAddTextINT4EntityLocations, CollectorAddTextINT4EntityFaceLocations, CollectorAddTextREAL8Locations, CollectorAddTextREAL8EntityLocations, CollectorAddTextREAL8EntityFaceLocations, CollectorAddLineLocations, CollectorAddPolyLineLocations, CollectorAddMeshLineLocations, CollectorAddTriangleLocations, CollectorAddMeshTriangleLocations, CollectorAddMeshTriangleNormalLocations, BitmapCreate, BitmapMarkForDeletion, SymbolCreate, SymbolMarkForDeletion, SymbolSetAddInScreen, SymbolAddPoints, SymbolAddLineStraight, SymbolAddLinePoly, SymbolAddLineCircle, SymbolAddLineBrick, SymbolAddLineCylinder, SymbolAddLineCone, SymbolAddLineSphere, SymbolAddFillTriangle, SymbolAddFillCircle, SymbolAddFillBrick, SymbolAddFillCylinder, SymbolAddFillCone, SymbolAddFillTubeBrick, SymbolAddFillTubeCylinder, SymbolAddFillSphere, Detach, and Attach to the feUserDefinedGraphics Object.
- Added PutAll to the feVector object.
- Added SnapToAxes, SetRotationAngles, and GetRotationAngles to the feViewOrient object.
- Updated GetLibrary and PutLibrary for the feFunction object.

The following functions have been added or updated:

- feFileReadIdeas
- feFileReadJT
- feFileWriteJT
- feSurfaceCornersMultiple
- feCurvesBreakAtIntersections
- feSurfaceExtract
- feGenerateCopy2
- feGenerateScale2
- feGenerateRadialCopy2
- feGenerateRotate2
- feGenerateReflect2
- feViewShow2
- feWindowShow2
- feViewShowMultiple
- feCreateFunction
- feAppModelInfoShow
- feAppSetModelInfoShow
- feSolidSlice2
- feSolidSliceAlongFace2
- feSolidSliceWithSheet2
- feSolidSliceWithCurve2
- feSolidExtractCenterlines
- feSolidAlignSplits
- feOutputGlobalPly

New and updated Global Variables

- Added Pref_ElemQualTetEPIA, Pref_ElemQualHexEPIA, Pref_ElemQualPenEPIA, Pref_ElemQualPyrEPIA, Pref_ElemQualTetEPIAVal, Pref_ElemQualHexEPIAVal, Pref_ElemQualPenEPIAVal, Pref_ElemQualPyrE-PIAVal, Pref_ShowAutoscale, Pref_SynchronizedRotation, and Pref_DefaultFreebodyReverse to set various preferences.
- Added Info_ViewShowAutoscale, and Info_FreebodyReverse to set various global variables.

Preferences

Views

- Added *Synchronized Rotation* option to *View and Dynamic Rotation* section. This option, which is disabled by default, allows you to synchronize the current dynamic rotation mode to the option currently specified for *Rotate Around* in *View, Rotate, Model* command. When enabled, the *View, Rotate, Rotate Around Coordinate System* mode will be selected and rotation will occur around the axes of the coordinate system currently specified for *Rotate Around* in *View, Rotate, Model*. To rotate around the screen axis when this option is enabled, specify “-1..Screen Axis” for *Rotate Around* in *View, Rotate, Model*.

User Interface

- Added *Autoscale* to *Show Entities Defaults* section, which will enable the *Autoscale* option by default for the *Window, Show Entities* command, as well as the *Show When Selected* functionality in the *Model Info* tree, *Data Table*, and *Connection Editor.*

Interfaces

- Removed the *Neutral Digits* option, as it is no longer needed. All Neutral files are now written using "Max Precision", which is 16 digits for double-precision real values, such as nodal coordinates, and 8 digits for single-precision real values, such as results.
- Updated titling of Groups created when Create Groups from INCLUDE files option is enabled and the absolute filename is longer than the allowable title length (79 characters). Previously the title was simply truncated to the filename with no path. Now, the full path is trimmed from the left on subdirectory boundaries to include as much of the path as will fit in the title.

Results

- Added *Reverse Values (New Model Default)* option to *Freebody Defaults* section. When the option is enabled, the *Reverse Freebody Values* option will be enabled by default for all new models. If opening an existing model, this option does not change the current setting of *Reverse Freebody Values*.

What's New for version 11.3.2

Geometry Interfaces

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

FEMAP Interface	Latest Supported Version
Solid Edge	Solid Edge with Synchronous Technology 9
NX	NX 11.0

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

Element Update

- Updated Modify, Update Elements, Line Element Orientation to be able to update the orientation of Spring/Damper Elements, which do not have coincident nodes, using the "Equivalent Vector Orientations" and "Make Perpendicular" options. "Equivalent Vector Orientations" will work when the original orientation was specified using the "CSys", "Node", or "From Property" option.

Analysis Program Interfaces

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

- NX Nastran Interface

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

NX Nastran Interface

- FEMAP with NX Nastran bundle now includes NX Nastran 11.0.

OLE/COM API

The following functions have been added or updated:

- Added feFileReadCSVResults to allow import of comma-separated results files.
- Updated feFileAttachResults to include ability to attach to ABAQUS results files (*.ODB files), comma-separated results files (*.CSV), and FEMAP Neutral Output files (*.FNO)
- Updated feModifyOrient to add ability to specify an orientation vector in the coordinate system specified in the nNodeOrCSysID argument, via method = -1. Also, documented method = 5, which allows you to specify an Orientation Coordinate System, which is only used by Spring/Damper and Spring/Damper to Ground Elements.

What's New for version 11.3.1

User Interface

Data Surface Editor

Data Surface Editor

- Added warning and question in Output Map Data Surface to not recalculate corner data when the Elemental Centroidal data has been edited by the user.

Meshing

- Improved the performance of the Subdivision mesher when meshing surfaces with a large number of holes. Quad meshing one sample surface with 90+ holes improved from 685 seconds to 12 seconds to complete the mesh.

Mesh Associativity

- Improved the Modify, Associativity, Automatic command to automatically associate nodes that are positioned on shared boundaries between solids to be associated with both solids. This eliminates cases that prevented some elements on one side or the other of a boundary between "unstitched" sheets (or solids if the "Check Solid Containment in Multiple Solids" option was off) from remaining unassociated because some of their nodes were not associated to the solid that contained the elements.

Connections (Regions, Properties, and Connectors)

- Added ability to display normal vectors on the faces of solid elements specified in Connection Regions.

Loads and Constraints

- Added "Element Pressure at Corners" as target for Model to Model output mapping.
- Added error message regarding mapping of unsupported element types for Line and Rigid elements. Data from these element types is ignored.

Output and Post-Processing

- Updated the Model, Output, Expand Complex command to allow specifying a range like [0 to 180 by 180] to get the final phase angle. Previously, because the default was [0 to 360 by 360] and we did not want to get the final 360 phase since it was equivalent to 0, the command stopped before the final phase. Now it still skips the final one if the final phase is greater than or equal to 360, but computes it if the final phase is less than 360. (PR 7709896)
- Added support to read displacements in basic coordinate system (Nastran data blocks BOUGV1 and BOPHIG) when importing or attaching to Nastran op2 files.
- Updated error message when Femap tries to read an ODB file that is from a version of ABAQUS which is newer than the latest version we support.

Geometry Interfaces

- Added option to the CATIA V5 read dialog to optionally read the extended title information. Always reads if there was no automatic titles. If there was an automatic title, this can append the extended information.
- Improved reading of part titles when CATIA V5 files are imported. Now creates titles based on the embedded Part Number, Revision, Nomenclature, and Description information.

Analysis Program Interfaces

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

- Nastran Interface

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

Nastran Interface

- Added ability to read CELAS1 and set formulation rather than converting to CELAS2.

OLE/COM API

New and updated API Methods

- Added GetFaceNodes2 method to the API Element object that adds an option to retrieve the face nodes in the order they are used for elemental corner pressures. GetFaceNodes retrieves nodes around an inward facing face normal for solid elements. GetFaceNodes2 does the same unless you follow the corner pressure convention which is around an outward facing normal.
- Added the GetGeomPropArray method to the API element object. It returns the individual PropertyID, MaterialID, Element Type and Topology, CG Location, and Length/Area/Volume when appropriate for a specified set of elements.

The following functions have been added or updated:

- Added feGetElementFacesFromSet, which allows you to pass in a set object to limit the elements which will be considered when selecting element faces.
- Updated feEntityVisibility, feEntitySetVisibility, and feEntityGetVisibility API calls to support freebody entities.

What's New for version 11.3

User Interface

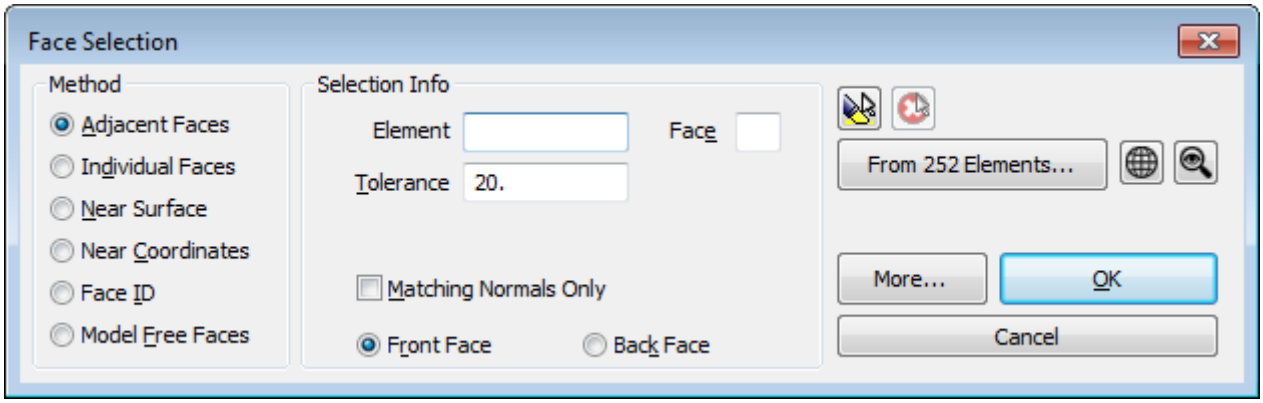
General, Menu, Toolbars, Model Info tree, Meshing Toolbox, PostProcessing Toolbox, Charting pane, Entity Editor, Connection Editor, Data Table

General

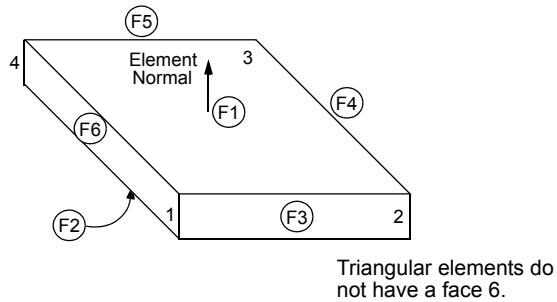
- Added *Add All Connected Elements* option to the *Pick^* menu of the standard entity selection dialog box when selecting elements.
- Added *on Element* to the *Method^* menu in the standard entity selection dialog box when selecting Coordinate Systems.
- Added *on Solid* to the *Method^* menu in the 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.



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*.

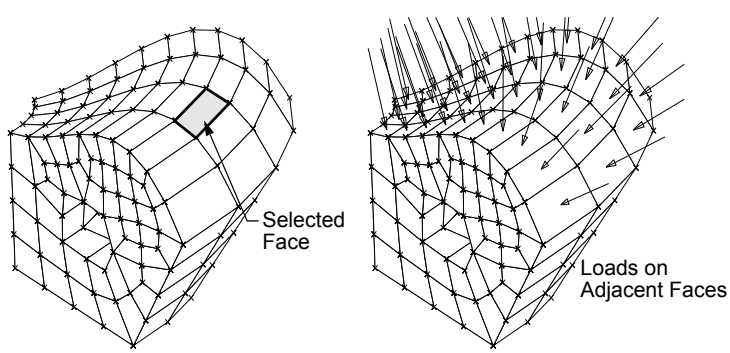
If you would like to limit which elements which will be considered when using a method which automatically determines faces (for instance, *Adjacent Faces*), click the *From (#) Elements...* button to use the standard entity selection dialog box to select elements. You may need to use the *Reset* button 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

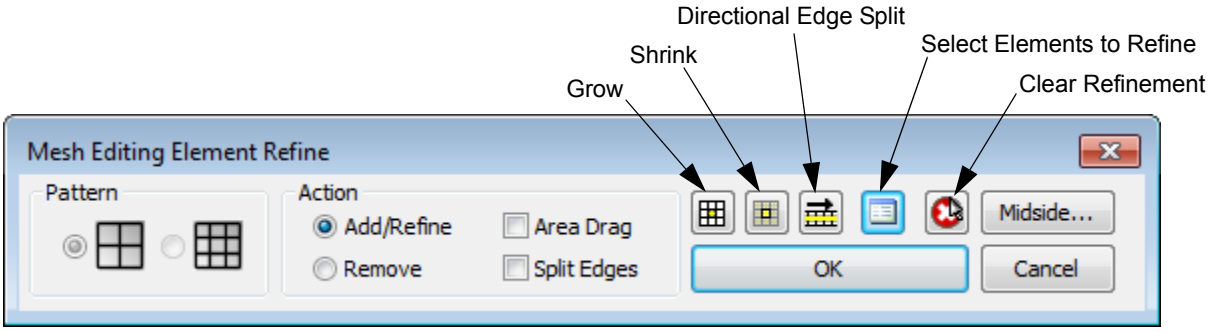
- Added *Plane to Plane* method for *Modify, Align...* commands to align geometry (*Point, Curve*, *Surface*, *Solid*, and *Volume*). When this method is used, the behavior is similar to *Between Coordinate Systems*, only each specified plane (X and Y axes) and each plane's normal direction (Z axis)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

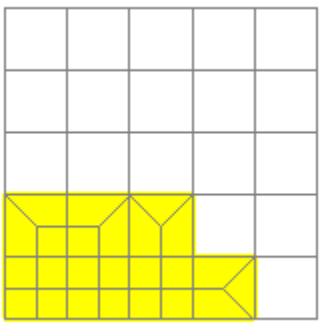


To use the command:

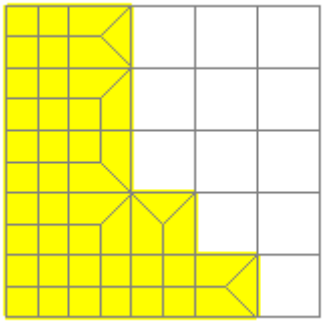
1. In the *Mesh Editing Element Refine* dialog box, pick a *Pattern* for the element refinement.
2. By default, *Action* will be set to *Add/Refine*. Move your cursor over the model, then "click and drag". As the cursor passes over an element, the command will preview the "split" on that element, as 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:

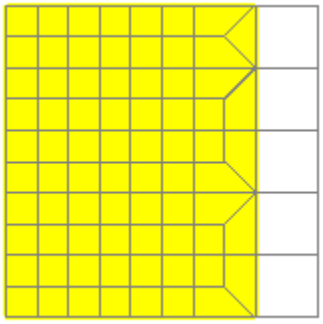
Area Drag - When this option is enabled, an attempt will be made to "expand" selection to all of the elements currently highlighted in a particular "area" of the model, when using "click and drag"



Original Refinement Area



Area Drag option disabled



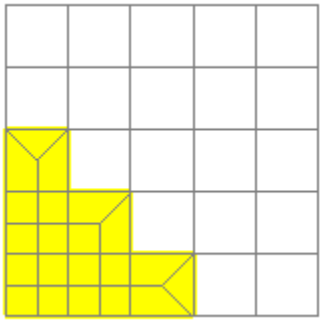
Area Drag option enabled

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..."

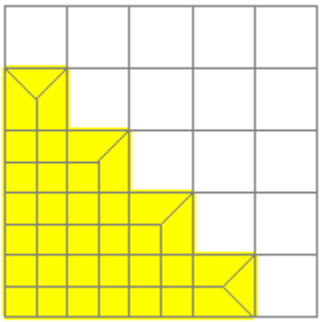
Icon buttons in the *Mesh Editing Element Refine* dialog box include:

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.

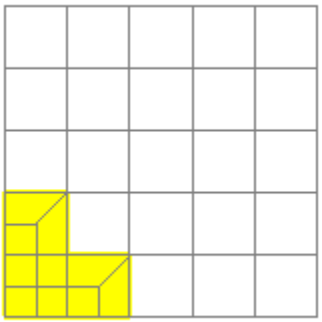
Shrink - Reduces current selection area(s) by removing one layer of elements from the outside of the current selection area(s). The *Area Drag* option is not considered when using the Grow functionality.



Original Refinement Area

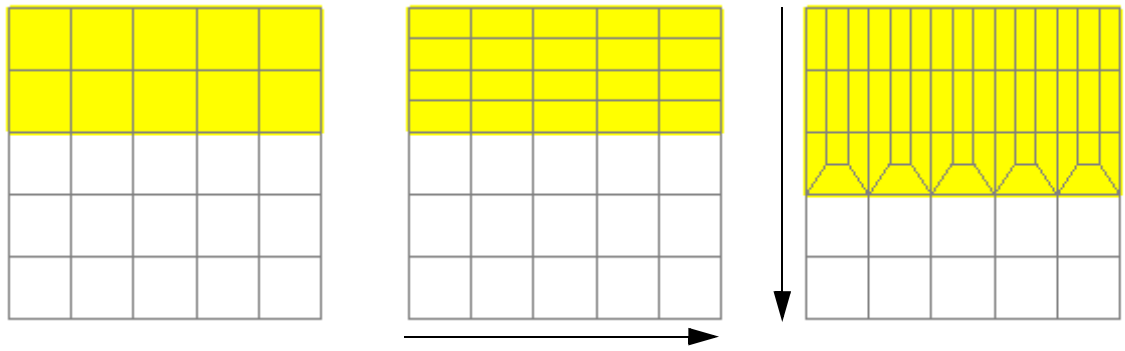


Area after single Grow



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*.

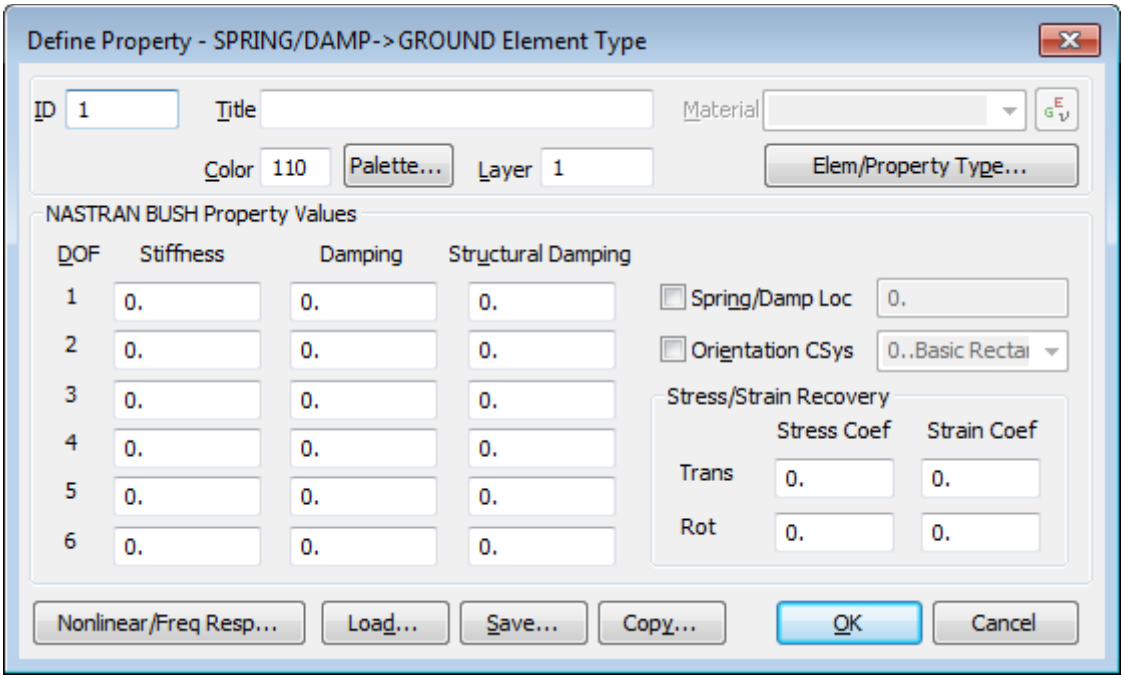
Elements

- 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.

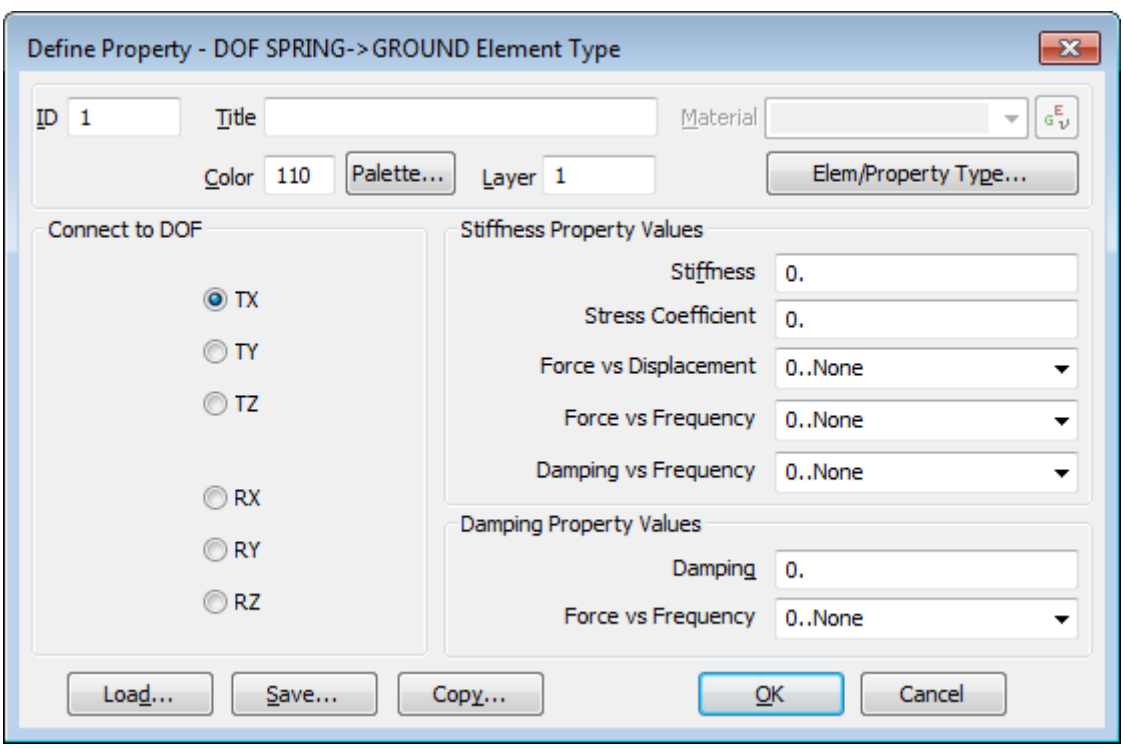
- Updated the name of the *Stiffness Matrix* element type to *General Matrix*. This type of element can now be used to include stiffness, damping, or mass between two nodes, using values in either a 6x6 or 12x12 matrix specified in the *General Matrix* property. Also, *General Matrix* elements are now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

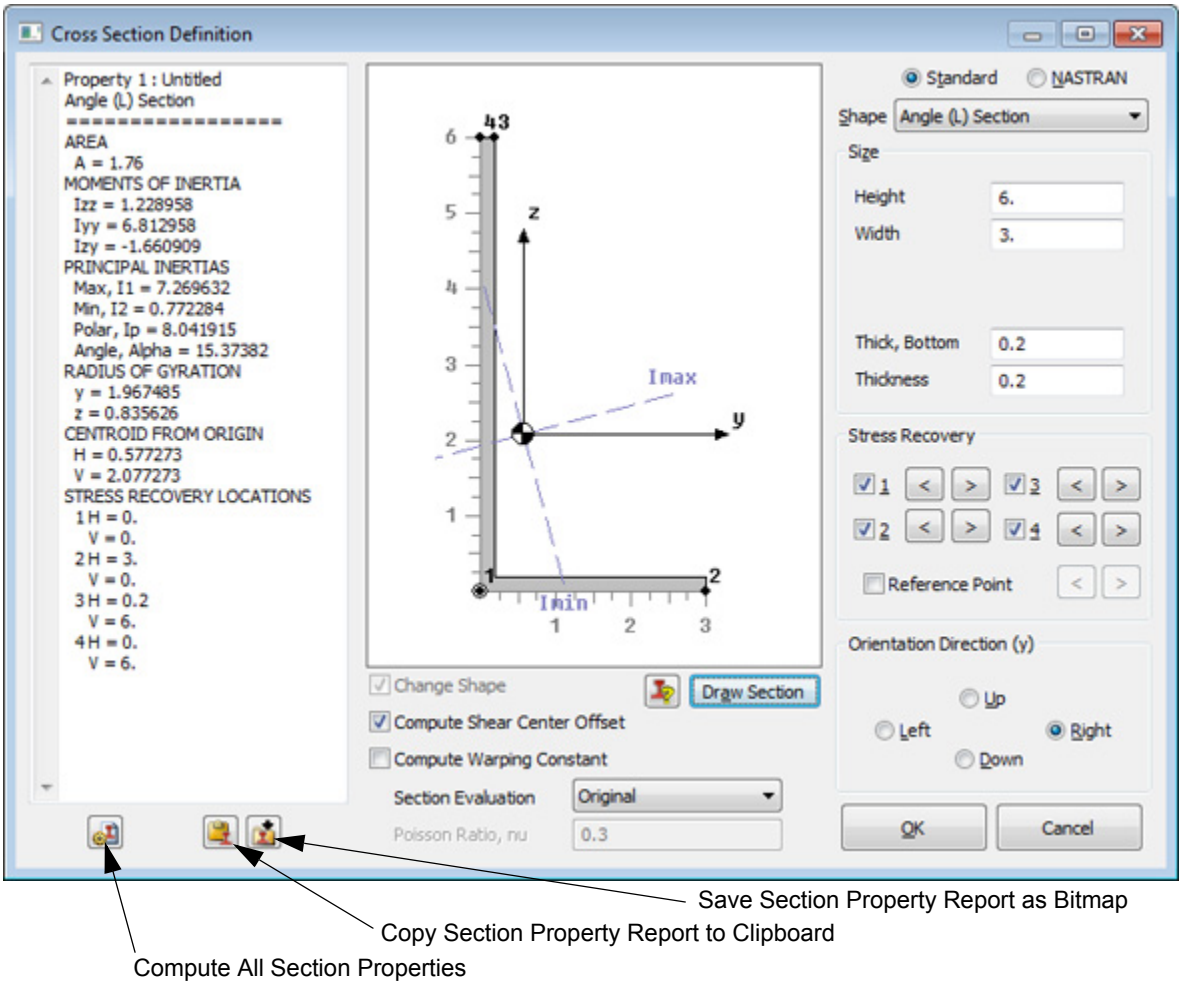


- Added *DOF Spring to Ground* property type which specifies *Stiffness*, *Damping*, *Connect to DOF*, and other options for the *Spring/Damper to Ground* element.



- 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.

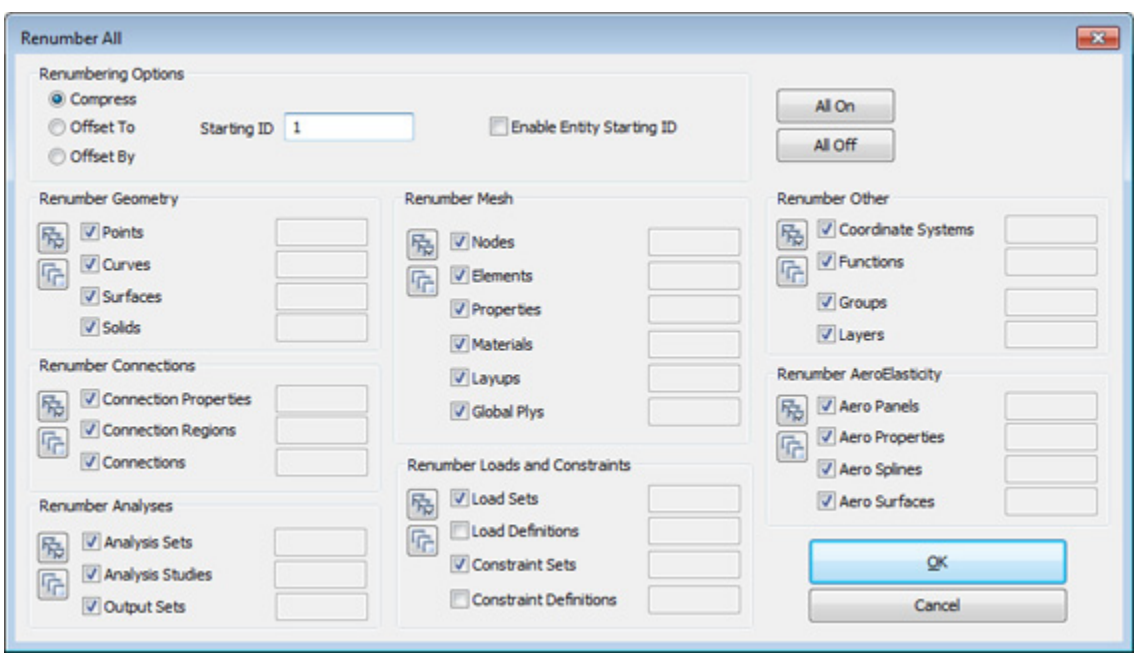
The *Copy Section Property Report to Clipboard* icon button will copy the picture of the defined cross section and the section properties currently displayed to the clipboard, while the *Save Section Property Report as Bitmap* icon button will use save the same information as a Bitmap file. It may be 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.

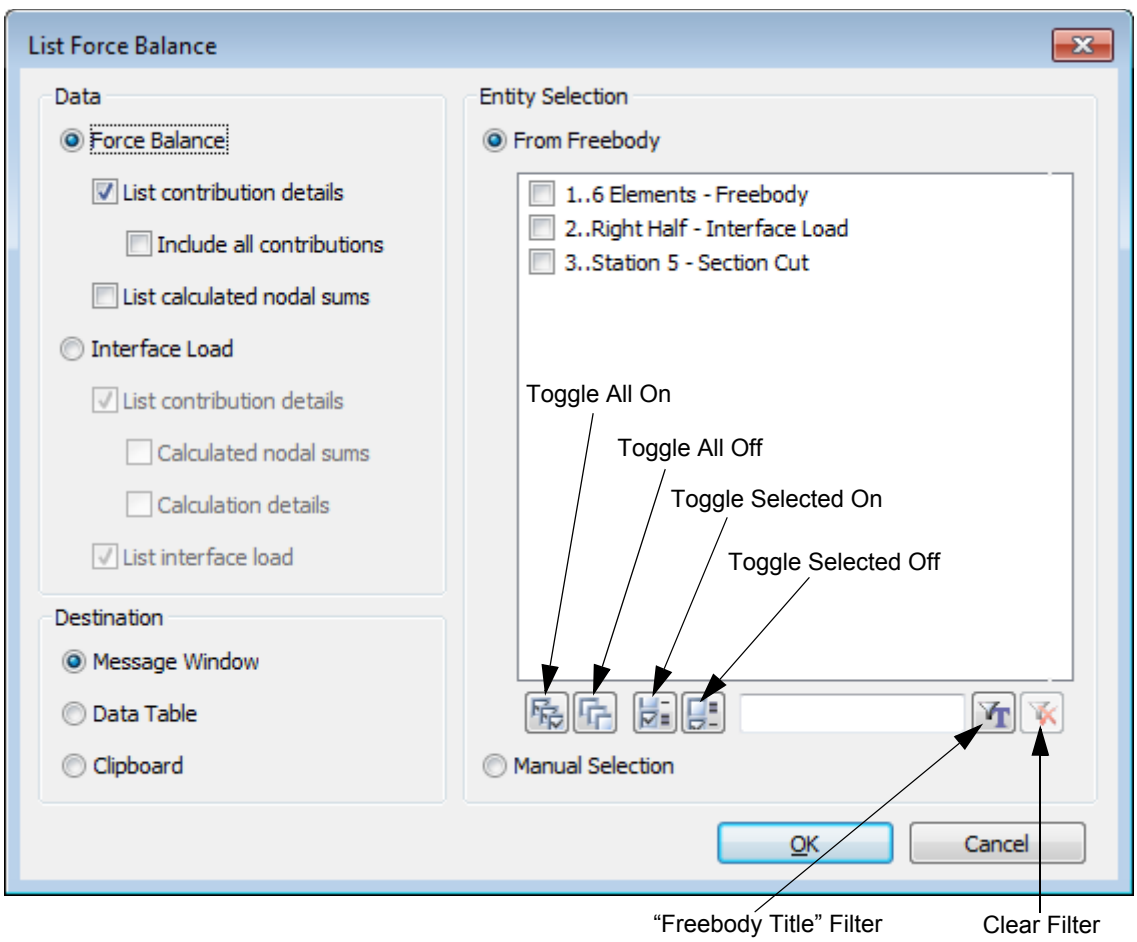
When the *Select Output from Contour Vector* option is disabled, then the output will be listed using a "Element ID" or "Node ID", a "Vector" (1, 2, or 3), a "Value", and "X" (always), "Y" (when output vector has 2 components), and "Z" (when output vector has 3 components).

- 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.

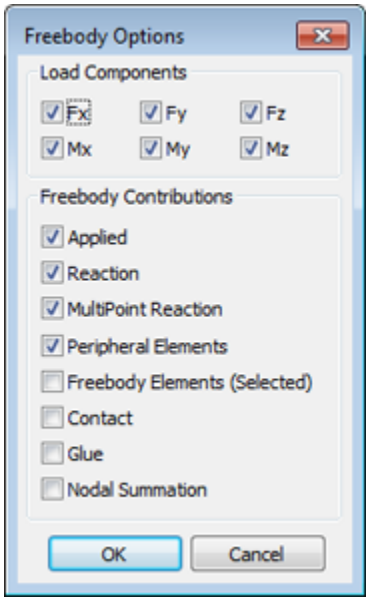
To reduce the number of entities in the list, simply enter text into the field to the left of the *Filter* icon button, then click the icon button. The list will be reduced to only those entries that contain the text you specified. You can now enter additional text, and press the *Filter* icon button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

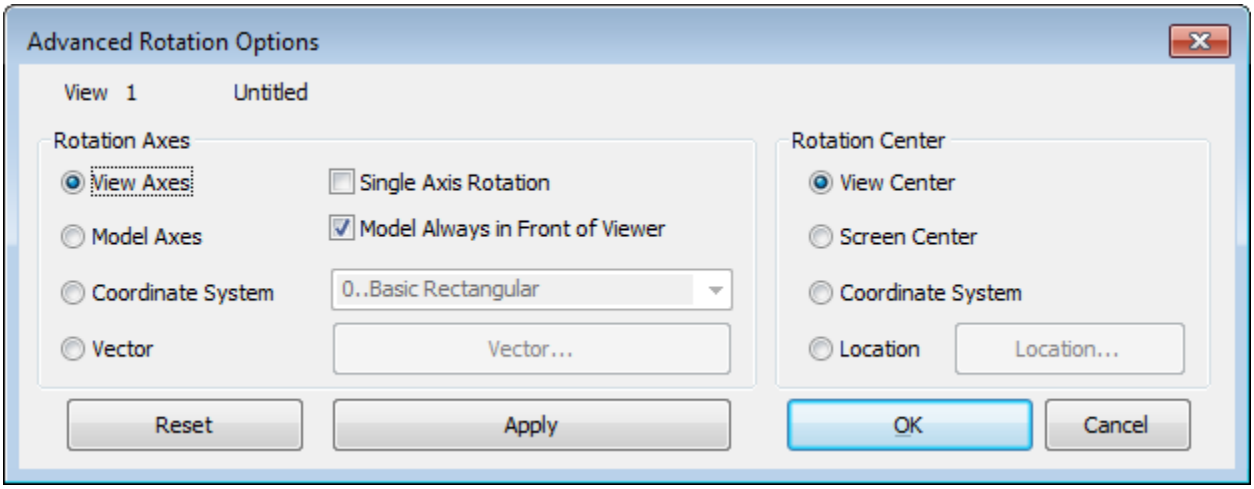
Note: The "Roll-Thru" functionality is designed to work appropriately only when the above settings are specified in the *Advanced Rotation Options* dialog box. If the settings for *Rotation Axes* or *Rotation Center* are changed, the model will still rotate and portions of the model will be still be "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.

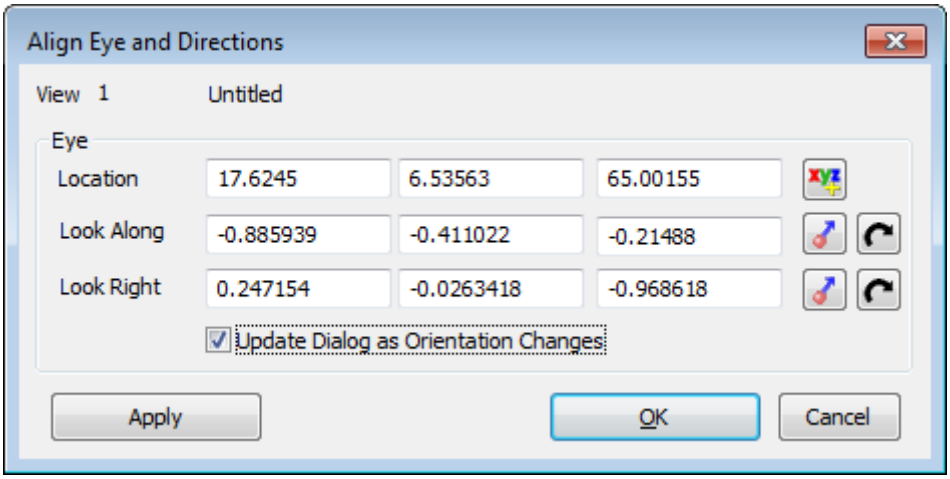
The "Single Axis Rotation" command can be also be accessed using *View, Rotate, Dynamic...* (Section 6.2.1.2, "View, Rotate, Dynamic..."). Use the *Options* button in the *Dynamic Display* dialog box to select this mode.

- 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

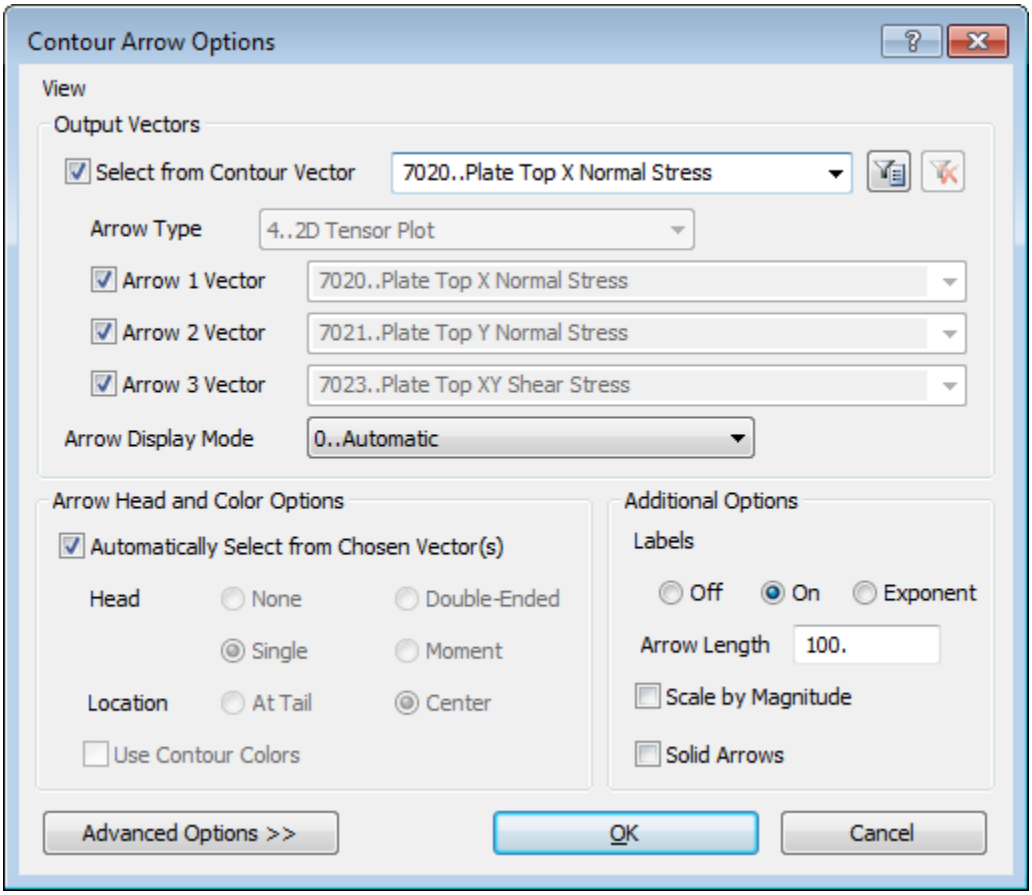
- Added ability to include Max/Min information from the currently displayed output vector in the *Post Titles*. To display the Max/Min information, use the *View, Options* command, set *Category* to *PostProcessing*, select *Post Titles* from the *Options* list, then set *Legend Style* to either "3..Titles 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 Option*s 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*.

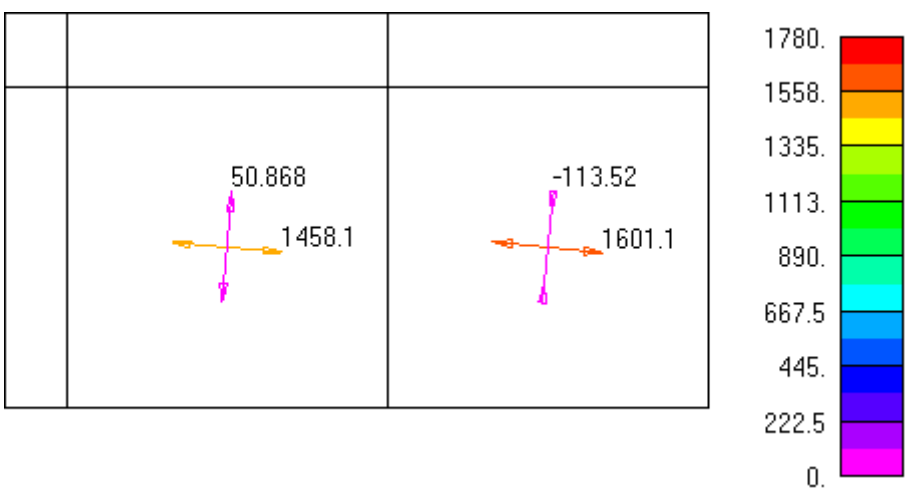
Note: In many cases, Arrow plots can be displayed using transformed output, which can be quite useful. When shell and solid output is not transformed, it is always displayed in the "current output orientation", which can be viewed in the *Current Output Orientation* dialog box. A single element shape 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.

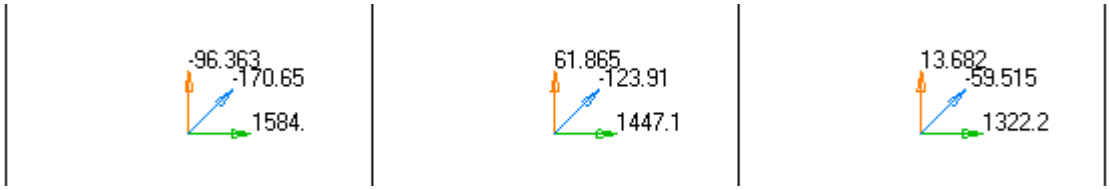
Single Value - Simply displays the output vector selected using the *Arrow 1 Vector* drop-down. You should also define a display direction by using the *Vector* button next to the *Arrow Type* drop-down, which allows you to select a vector using the standard *Vector Definition* dialog box. This *Arrow 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

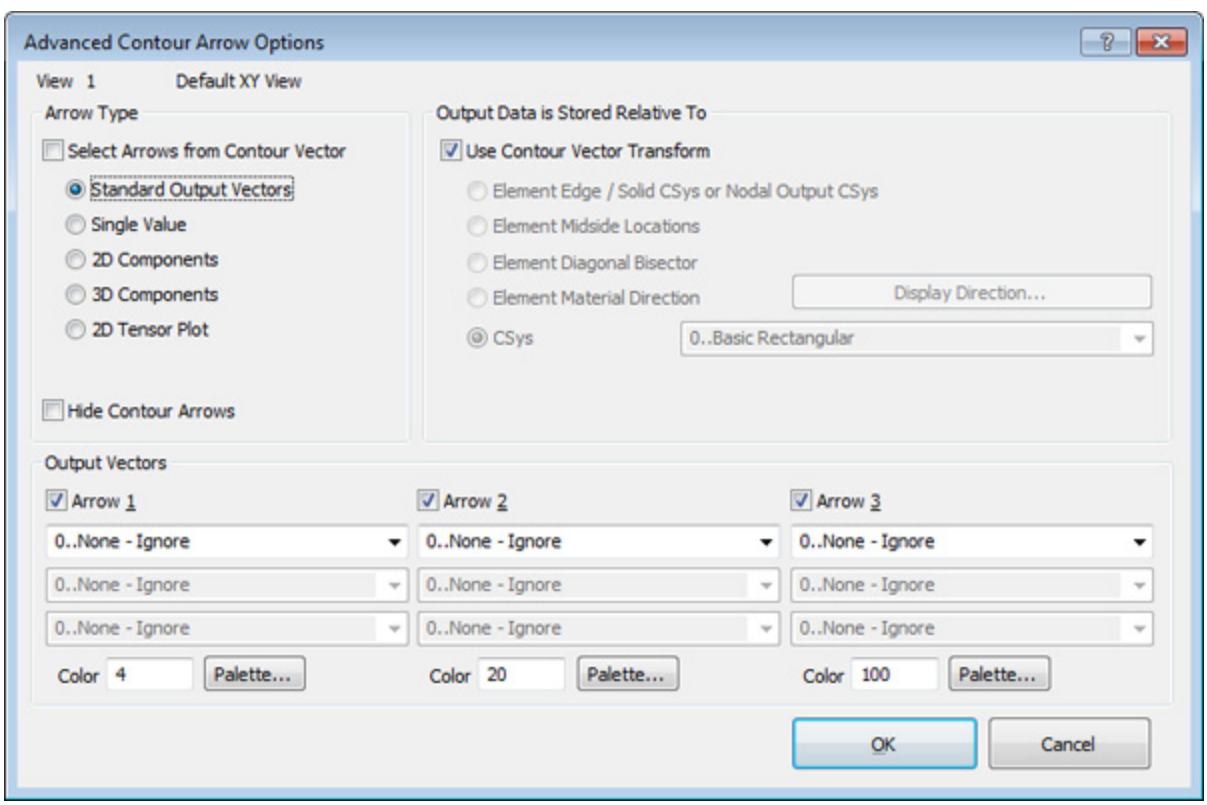
This button opens the *Advanced Contour Arrow Options* dialog box, which allows you to setup Contour Arrows using a "legacy" dialog box found in earlier versions of FEMAP.

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)

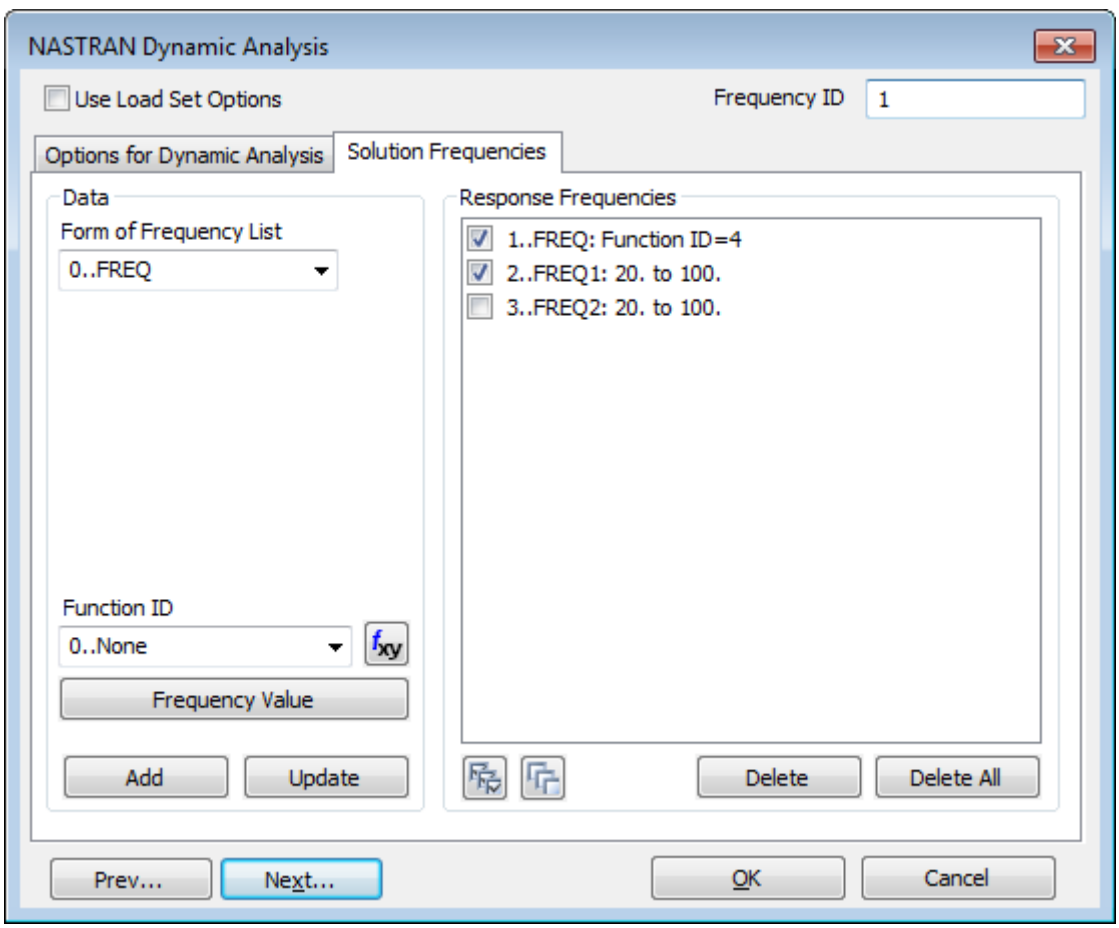
- Added *Additional Command Line Arguments* field to the *NASTRAN Executive and Solution Options* dialog box. This allows the user to include command line arguments which are not explicitly supported by FEMAP, when launching Nastran. If any command line arguments are specified in the *Arguments* field 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.

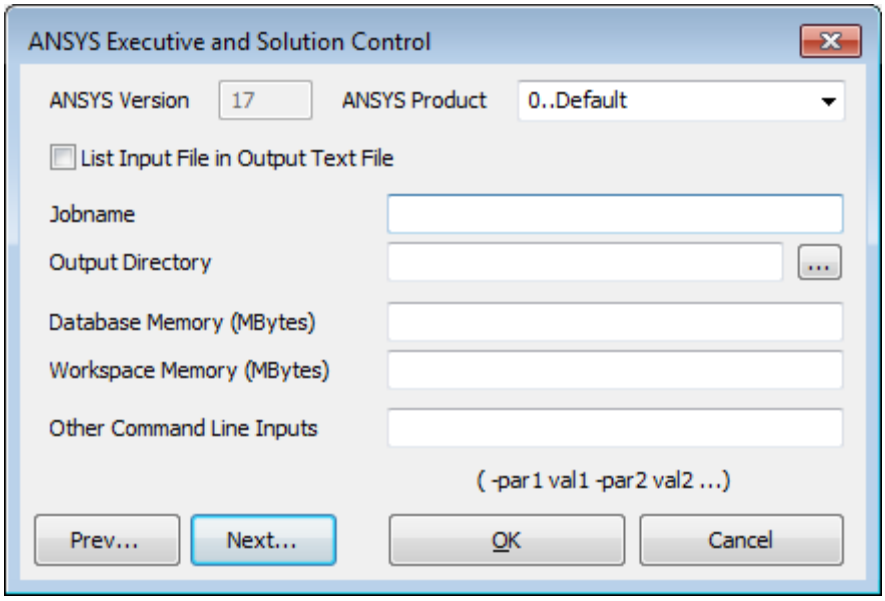
A number of bugs were corrected

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

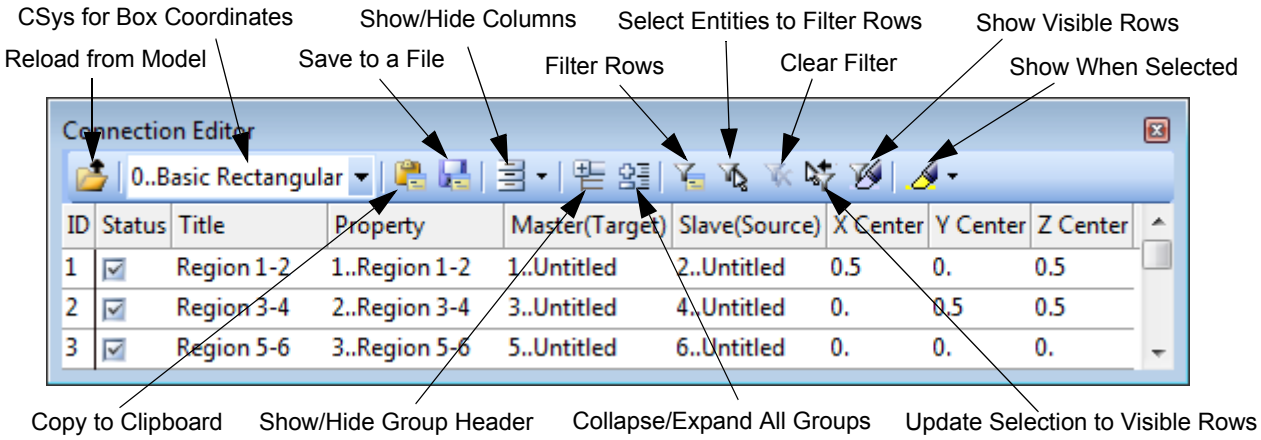
- Added support to read and write non zero constraints.

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

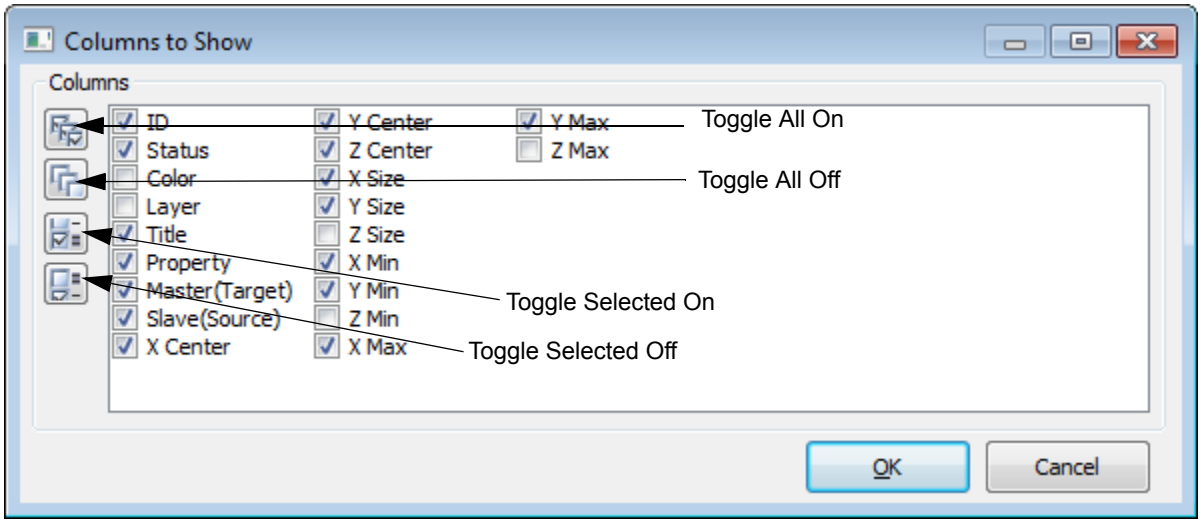
Note: Because of the Rich Text Format copied from the *Connection Editor*, you may need to do some resizing of columns in order to create a correctly formatted table when it is copied into the *Messages* pane. When copying tables from the *Connection Editor*, there should be no issues when pasting into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

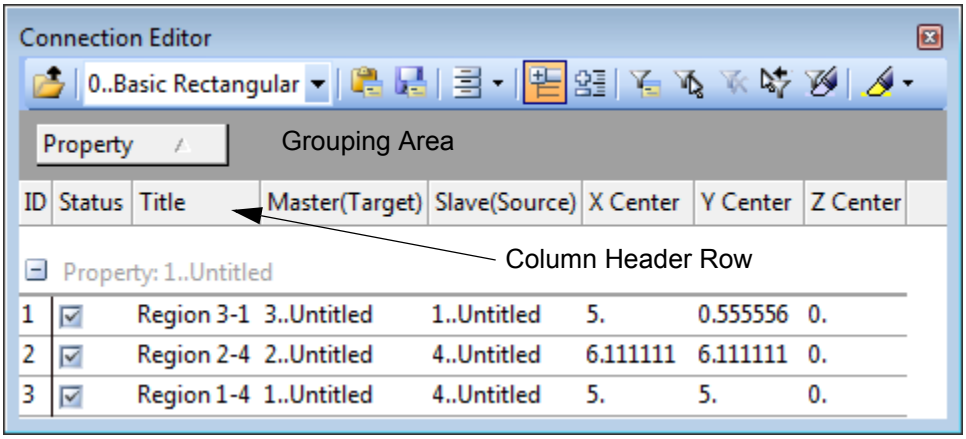
H**ide/Show Columns Menu** - Clicking this icon brings up a menu with commands and a listing of all available columns (visible columns are marked with a check mark in an orange box). By highlighting a column and clicking the mouse, you can toggle the column from hidden to visible and vice versa. This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

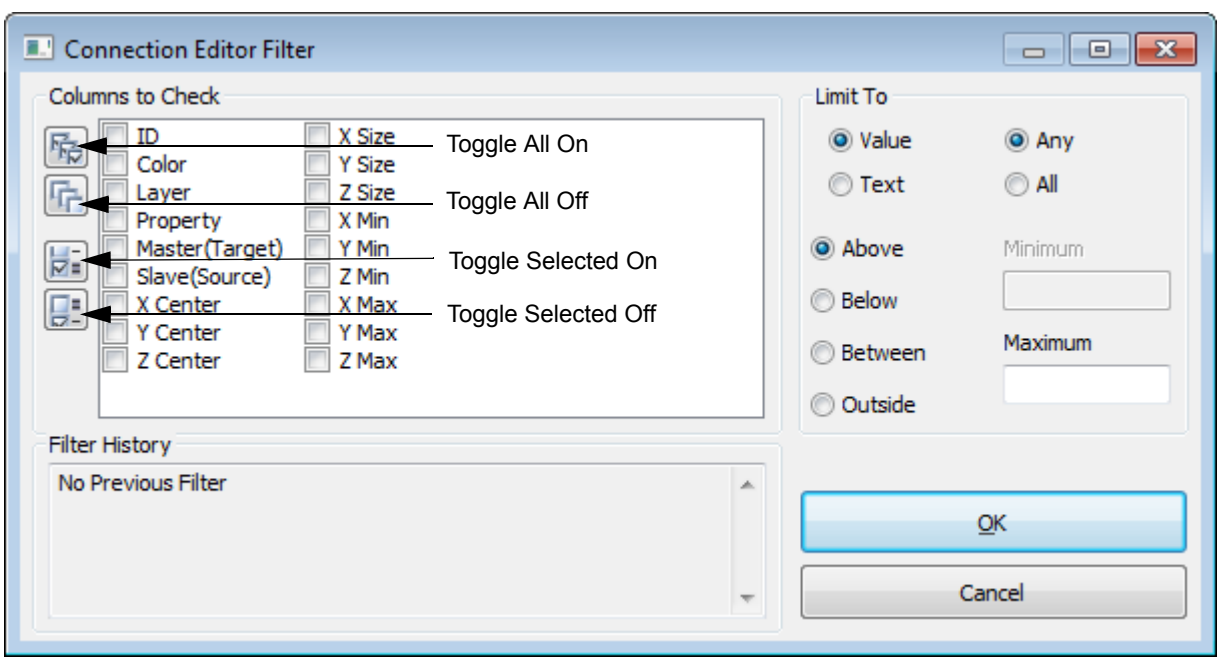
nector data using a specific group header or set of group headers. This allows you to "group" Connectors in the table with similar characteristics (such as the same connection *Property*, *Master (Target)* region, *Color*, etc.) together for a specific purpose. The grouping is multi-level, therefore 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, the region designated the *Master (Target)* becomes the *Slave (Source)* and the region designated the *Slave (Source)* becomes 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*.

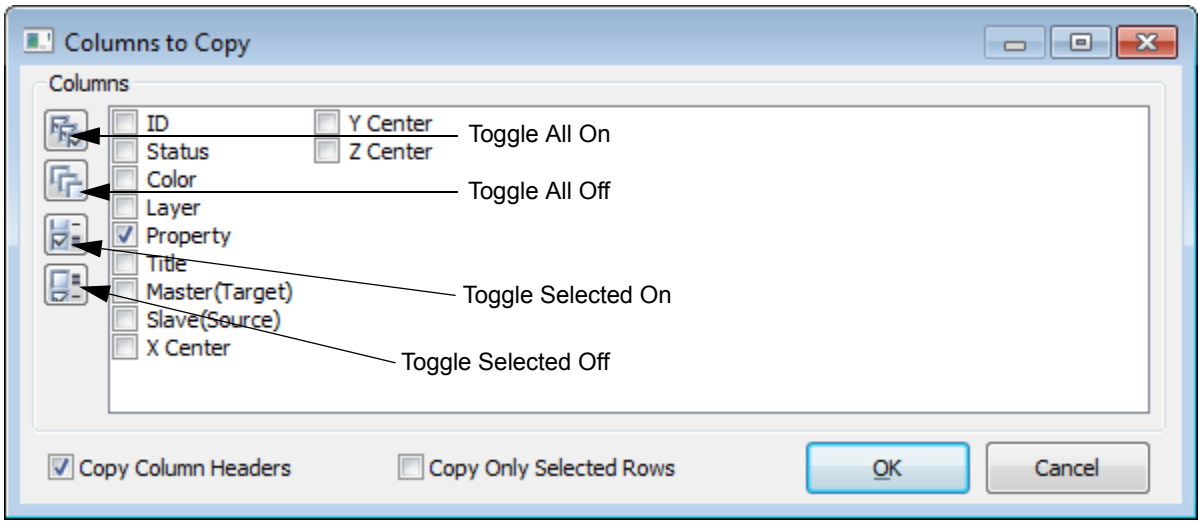
Note: It is possible to choose multiple entity rows at the same time and perform operations on all of the rows at once. In order to choose multiple entity rows, hold down the CTRL key and select individual entities with the mouse or hold down the SHIFT key and select a first row and a last row and all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

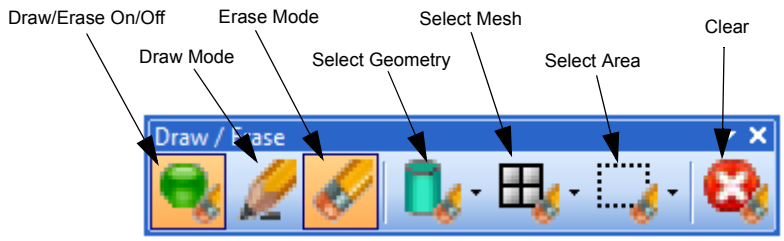
Context Sensitive Menu (Other) - When the cursor is not positioned over a column header or a row, right mouse clicking will bring up a context sensitive menu. This menu includes the *New Connector*, *New Connection Region*, *New Connection Property*, and *Delete All* commands from the context-sensitive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

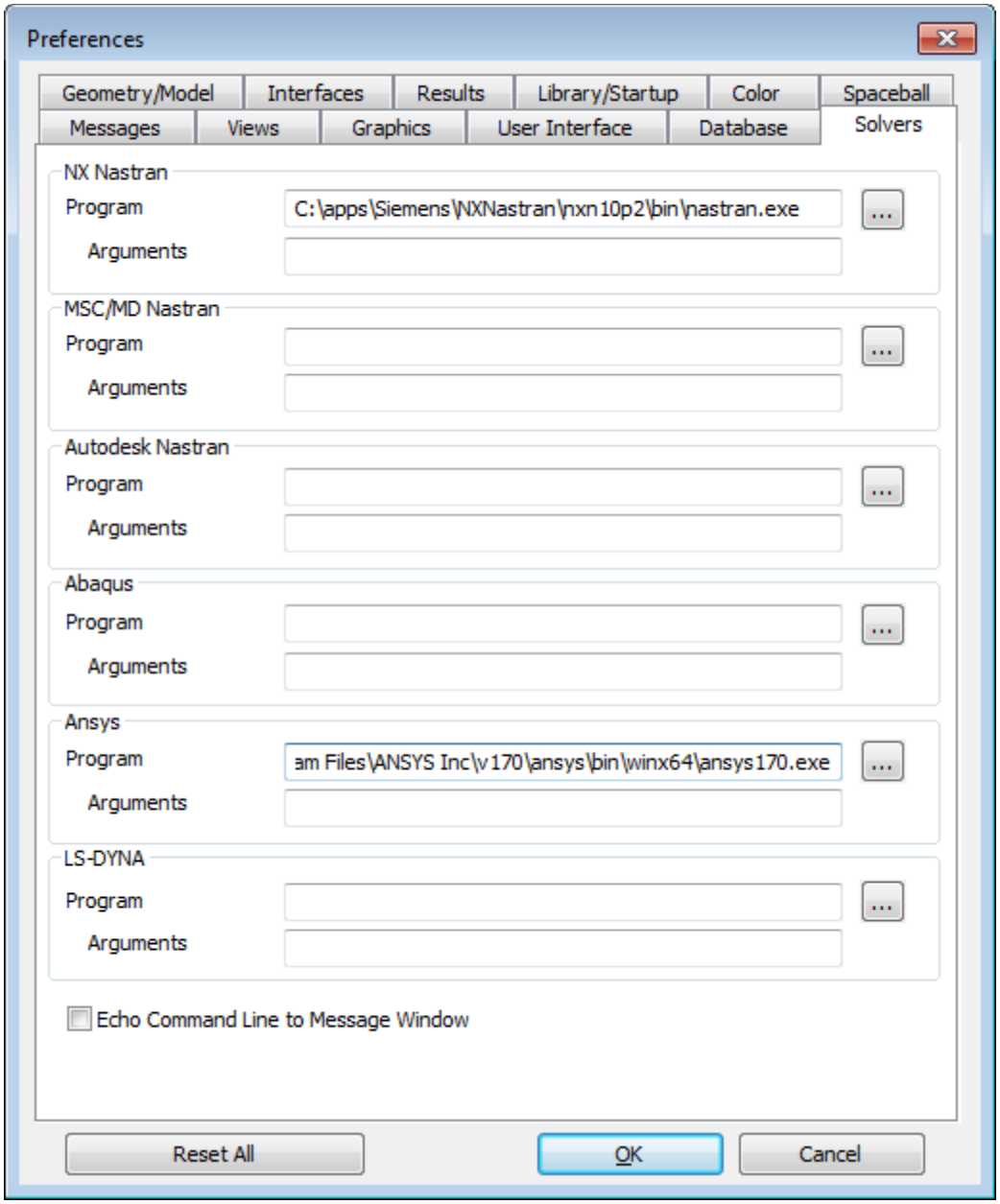
- Added *Reset Next ID after Delete All* to *Database Options* section.

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.

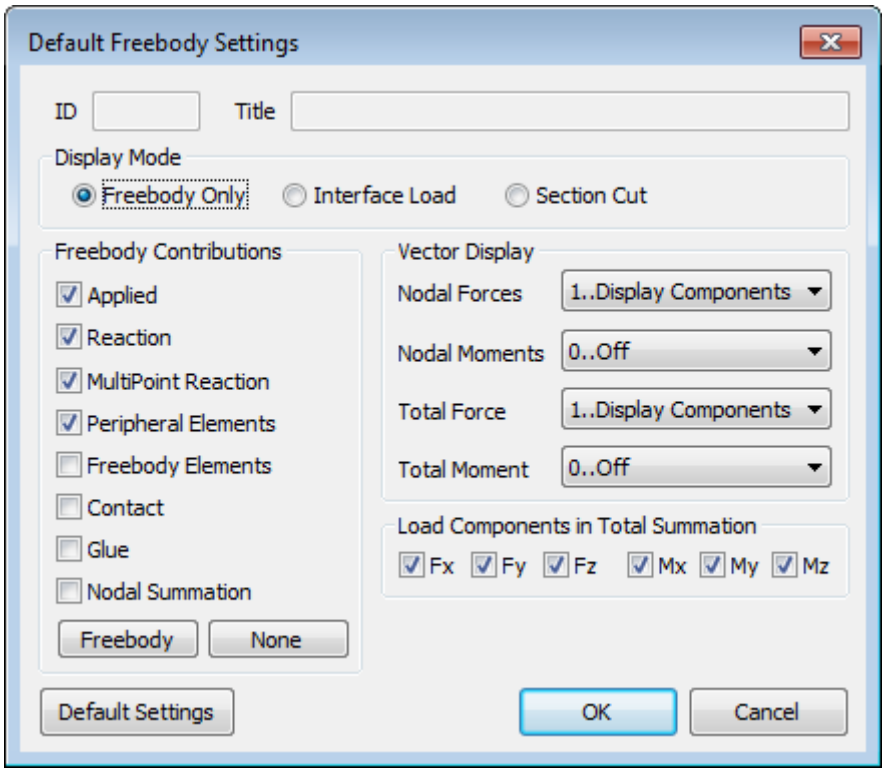
When this option is enabled, a Nastran Subcase ID will be stored as the "Case" ID on each Output Set created by importing or attaching to a Nastran Results File. If another Nastran Output File containing the same Nastran Subcase ID is imported or attached, the appropriate "Revision" number will also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