What's New in FEMAP 11.4

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

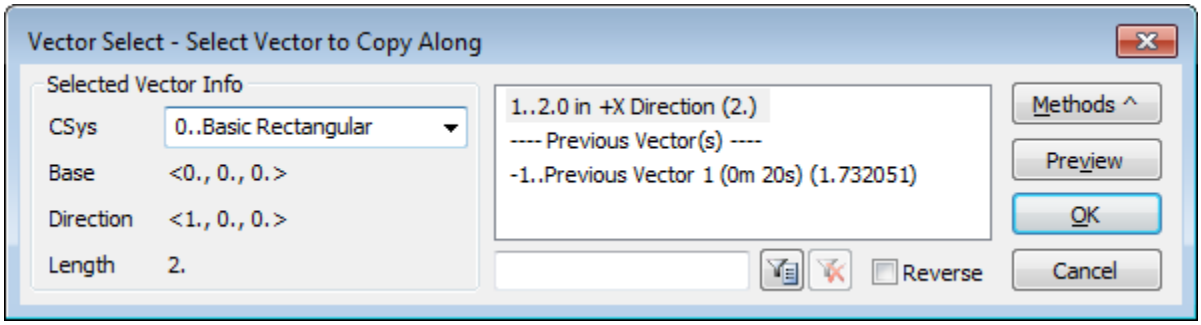
What's New for version 11.4

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

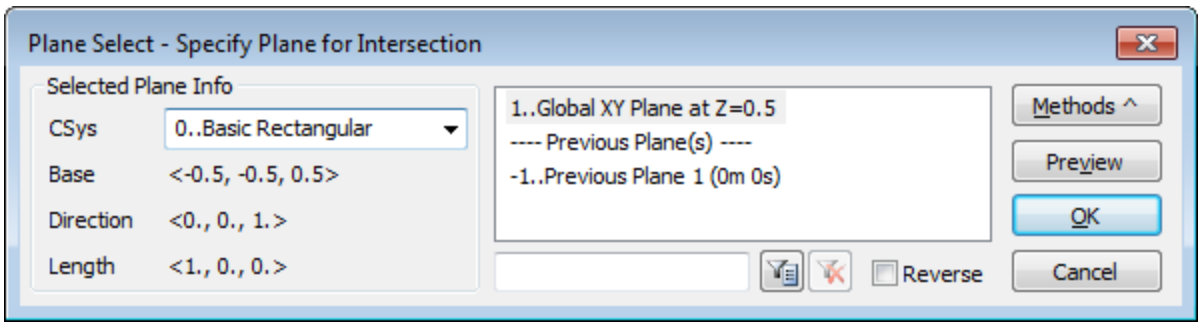
- Added *Tetra EPIA*, *Pyr EPIA*, *Penta EPIA*, and *Hex EPIA* as options when using the *Model Data Value* option for *Pick^* in the standard entity selection dialog box.
- Added *Saved* option to the *Methods^* menu on the standard *Vector* definition dialog box.



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

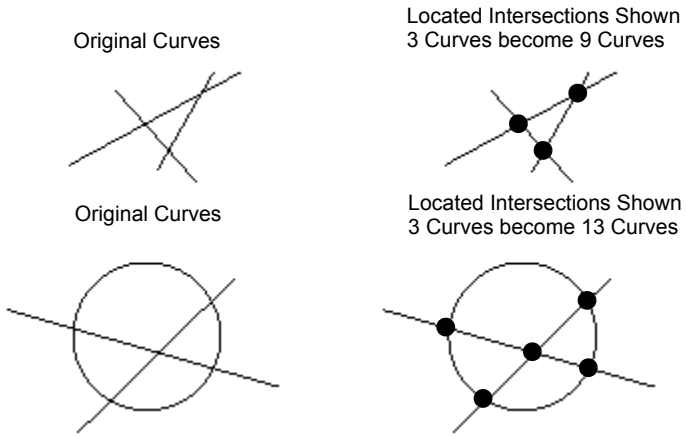
- Added *Modify, Break, At All Intersections* command.

... 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.

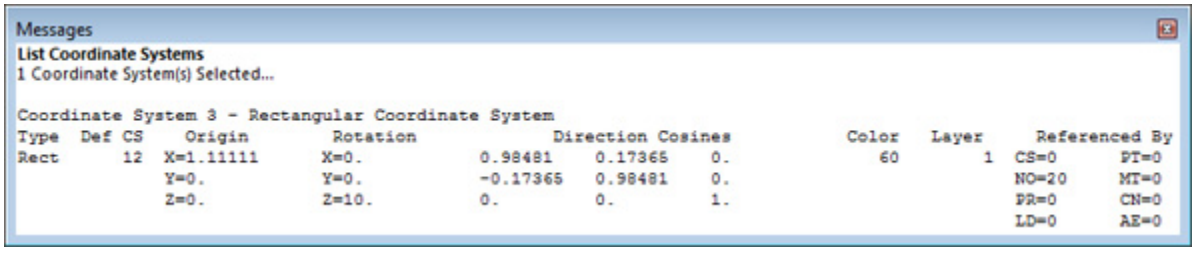
Connections (Regions, Properties, and Connectors)

- 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:



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*.

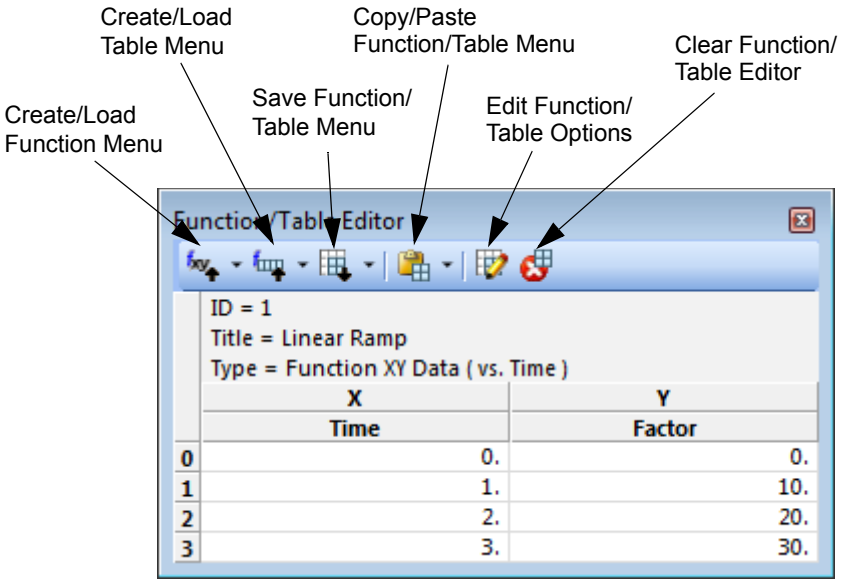
Tools

- 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:

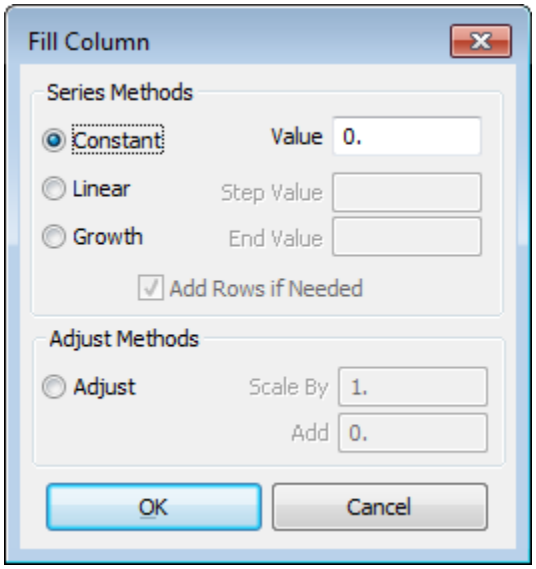
Copy Selected - copies the values in the highlighted group of cells to the clipboard.

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*.

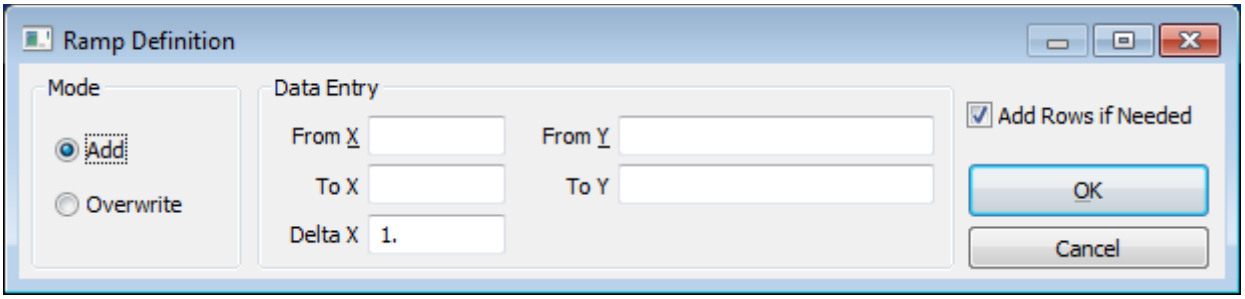
Row Height - allows you to enter a height, in pixels, for all rows in the *Function/Table Editor*.

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

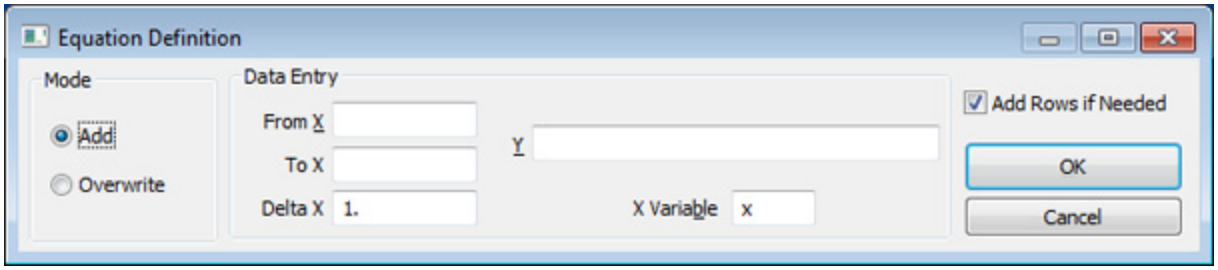
When a function is loaded in the *Function/Table Editor*, clicking the right mouse button will cause the context sensitive menu to appear, which contains commands designed to facilitate creation of functions.

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.

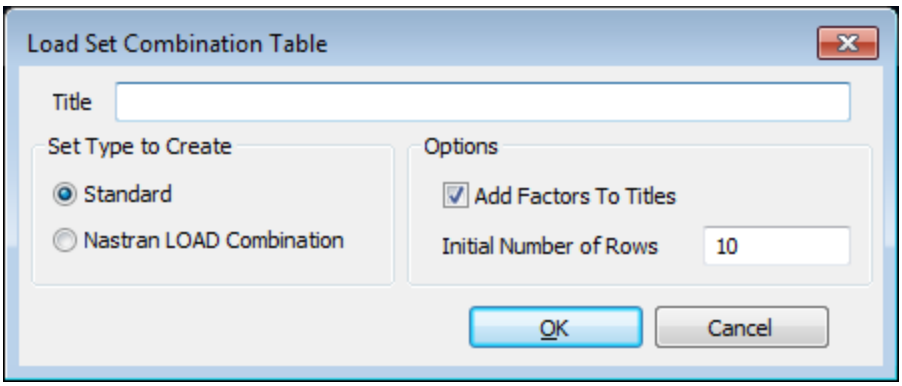


When *Mode* is set to *Add*, values can be added to empty cells or new rows can be added to the end of an existing function. When *Mode* is set to *Overwrite*, cells in an existing function will be replaced with new values. Similar to *Ramp...*, enter values for *X*, *To X*, and *Delta X*. However, for *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:



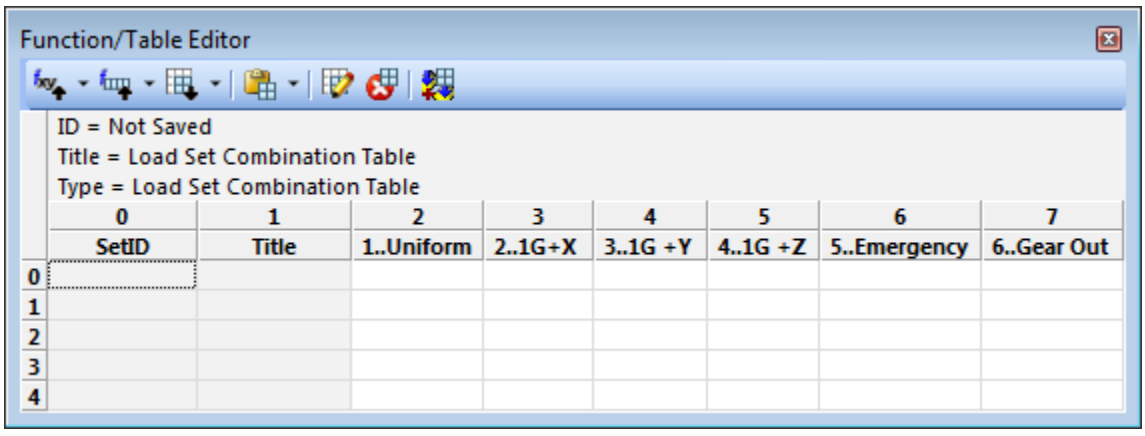
Use the *Set Type to Create* option to choose the type of load set(s) to create, *Standard* or *Nastran LOAD Combination* load sets. See Section 4.3.1, "Model, Load, Create/Manage Set..." for more information on *Load Set Type*. Combinations of existing *Nastran LOAD Combination* load sets can be created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

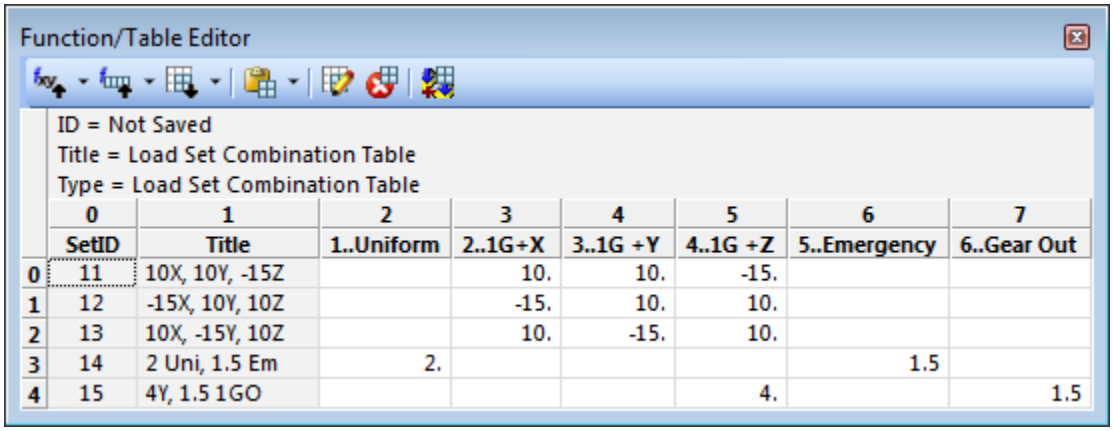
Additional commands on the context-sensitive menu for row headers:

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

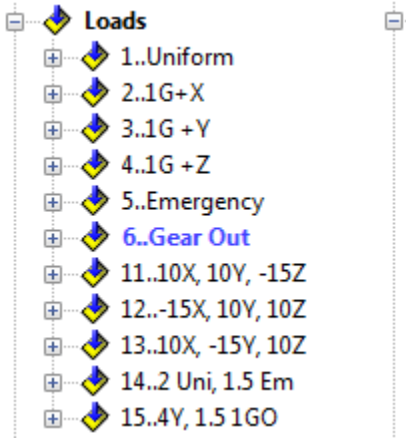
This type of table can be used to create 5 new load sets from 6 existing load sets. Simply enter scale factor values into various cells (optionally, a specific *SetID* and *Title* can be entered for each row):



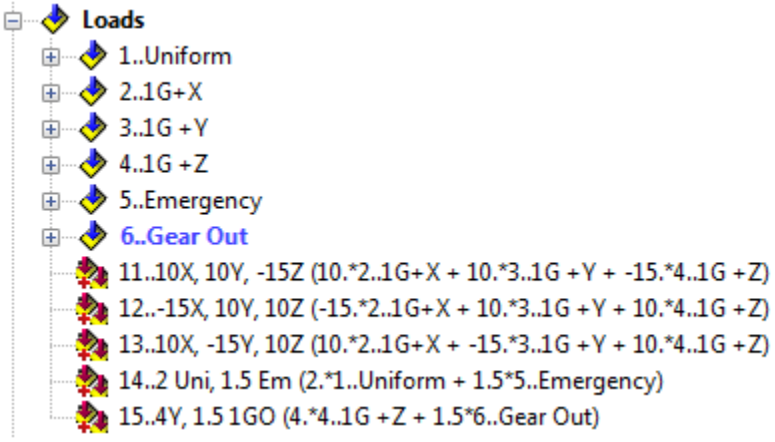
	0	1	2	3	4	5	6	7
	SetID	Title	1..Uniform	2..1G+X	3..1G +Y	4..1G +Z	5..Emergency	6..Gear Out
0	11	10X, 10Y, -15Z		10.	10.	-15.		
1	12	-15X, 10Y, 10Z		-15.	10.	10.		
2	13	10X, -15Y, 10Z		10.	-15.	10.		
3	14	2 Uni, 1.5 Em	2.				1.5	
4	15	4Y, 1.5 1GO				4.		1.5

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



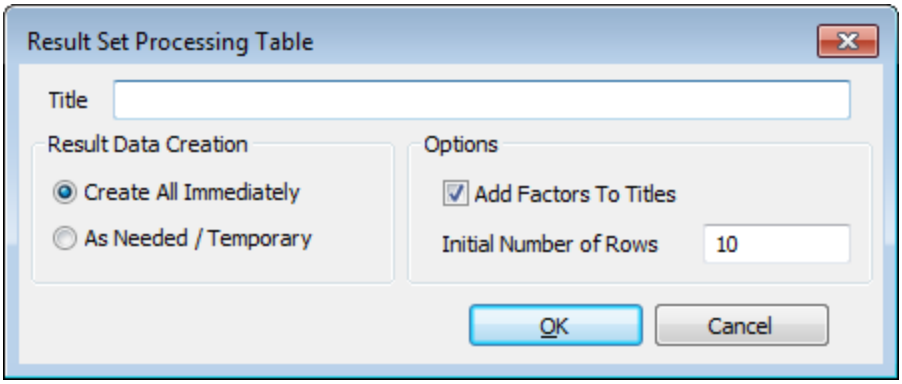
Titles Defined
Add Factors To Titles = On
Set Type to Create = Nastran LOAD Combination



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:



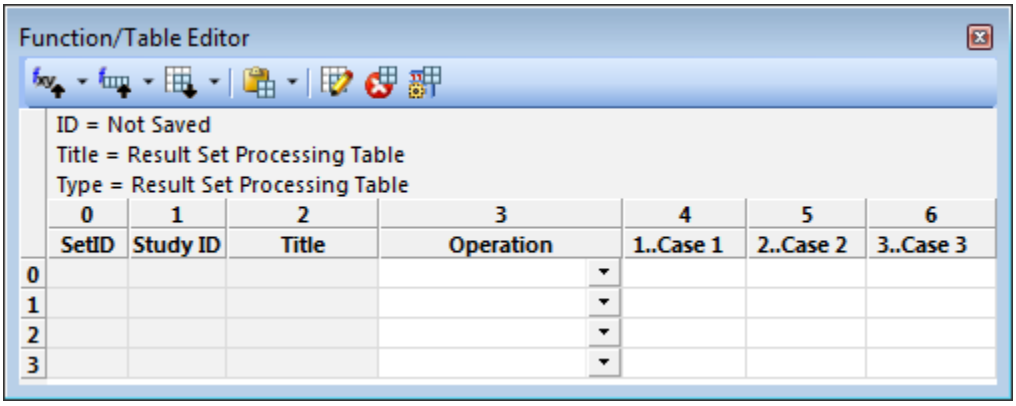
Use the *Results Data Creation* option to choose the method for creating the output set(s), *Create All Immediately* or *As Needed/Temporary*. When using *Create All Immediately* the output values will be stored in the created or updated output set(s). Output sets created with this option will be displayed 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.

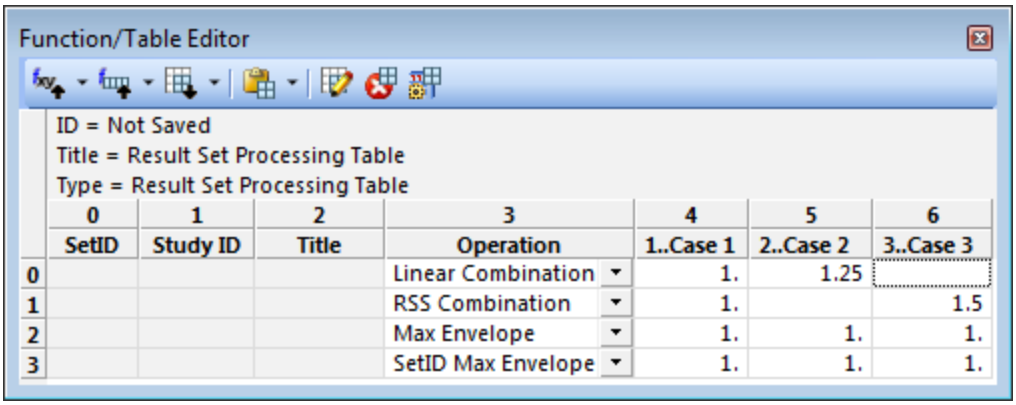
Add Result Sets - allows you to add columns by selecting any number of output sets using a dialog box

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

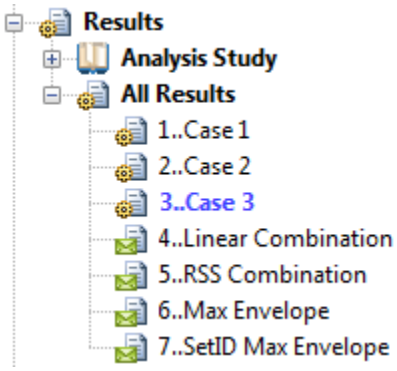
This type of table can be used to create 4 new output sets, each using a different Operation (a *Linear Combination*, a *RSS Combination*, a *Max Envelope*, and a *SetID Max Envelope*), using 3 existing output sets. Simply enter scale factor values into various cells. It is typical to simply use 1.0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



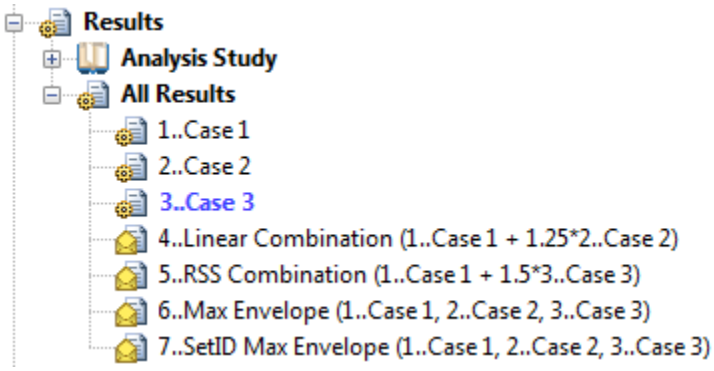
	0	1	2	3	4	5	6
	SetID	Study ID	Title	Operation	1..Case 1	2..Case 2	3..Case 3
0				Linear Combination	1.	1.25	
1				RSS Combination	1.		1.5
2				Max Envelope	1.	1.	1.
3				SetID Max Envelope	1.	1.	1.

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.

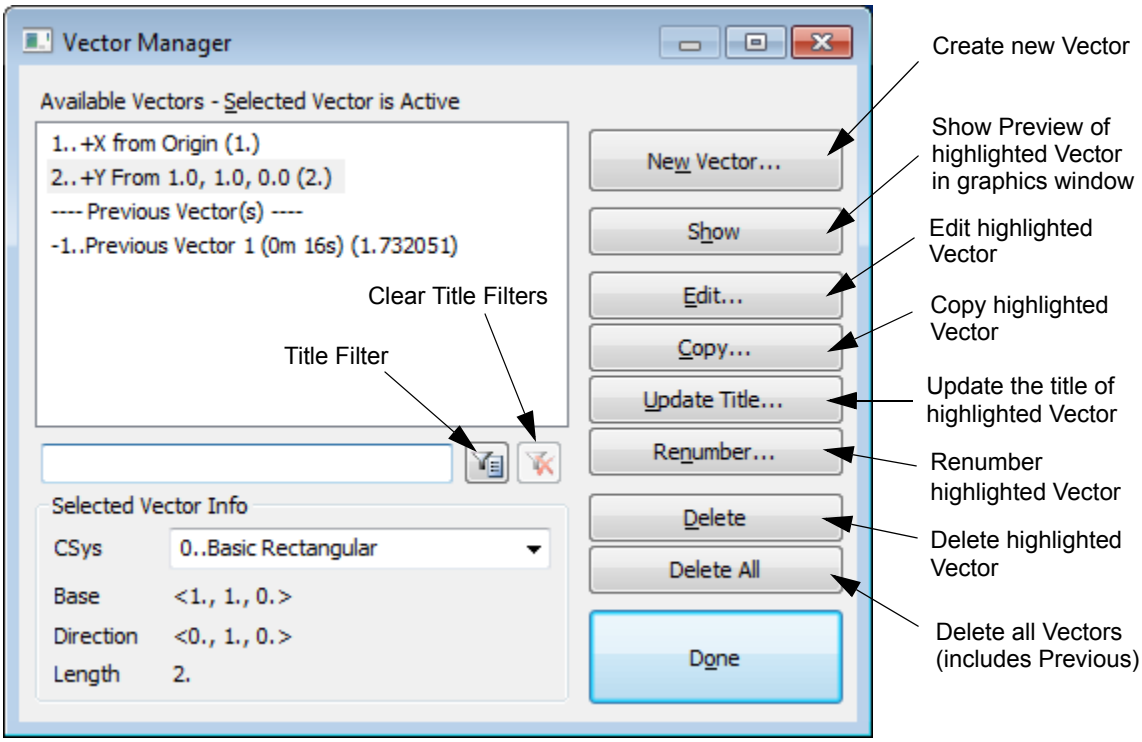
- Added *Tools, Vector Manager...* command.

... 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

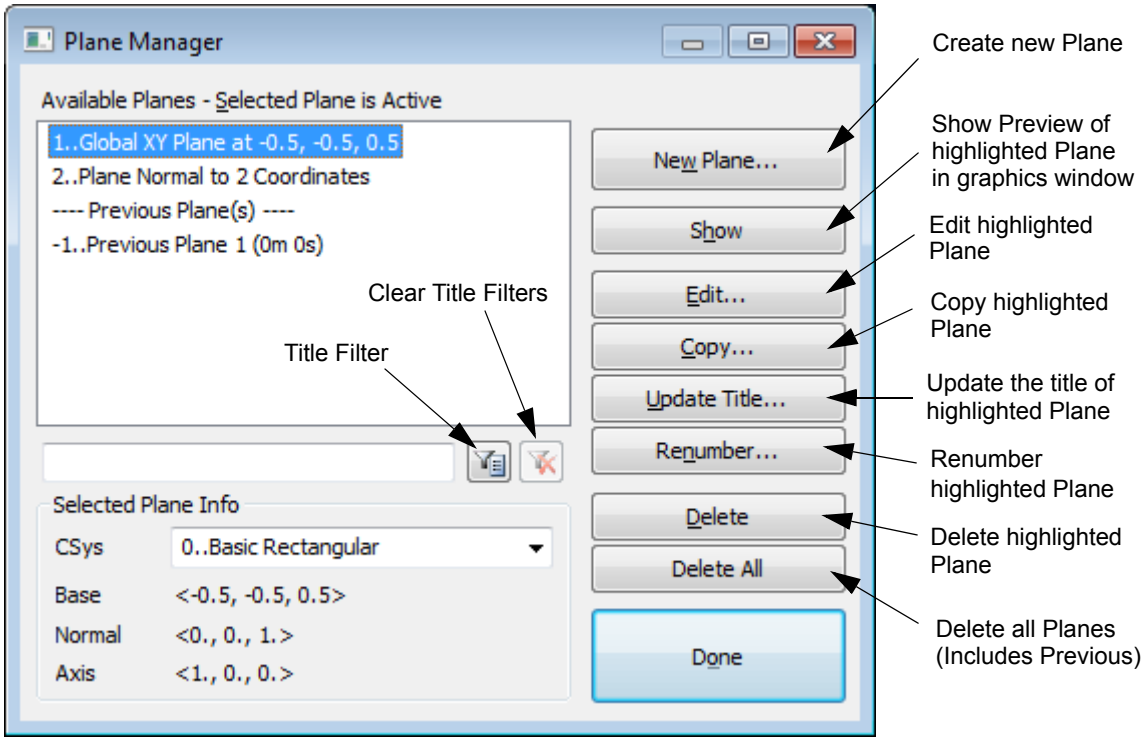
To reduce the number of vectors being displayed in the *Available Vectors...* list, enter text into the field below the list, then click the *Title Filter* button and only those vectors that contain the specified text will remain in the list. If additional text is entered and the *Filter* icon button 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.

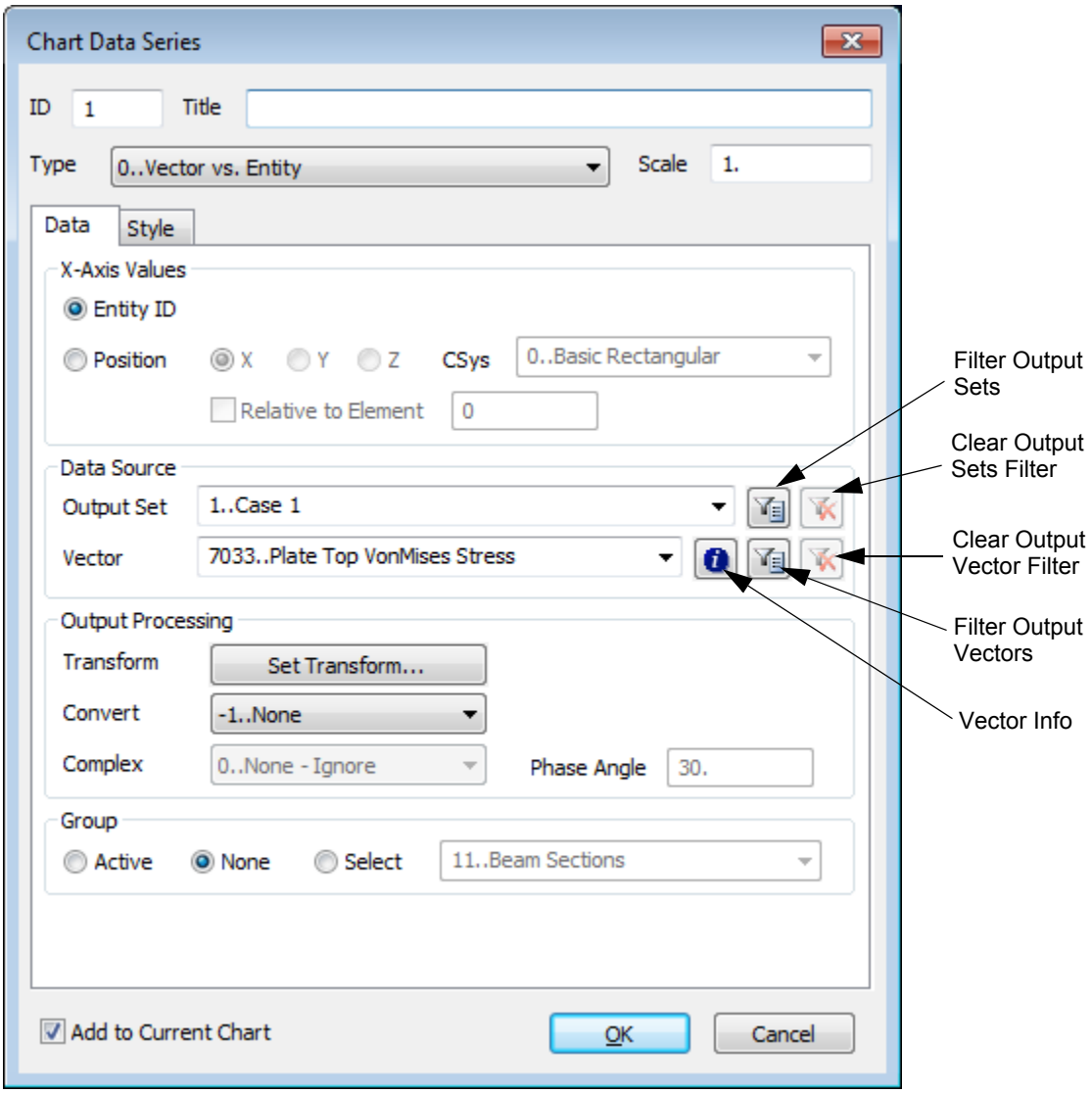
Finally, *Delete All* will delete all of the "saved" planes and any planes currently in the *Previous Plane(s)* section.

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.

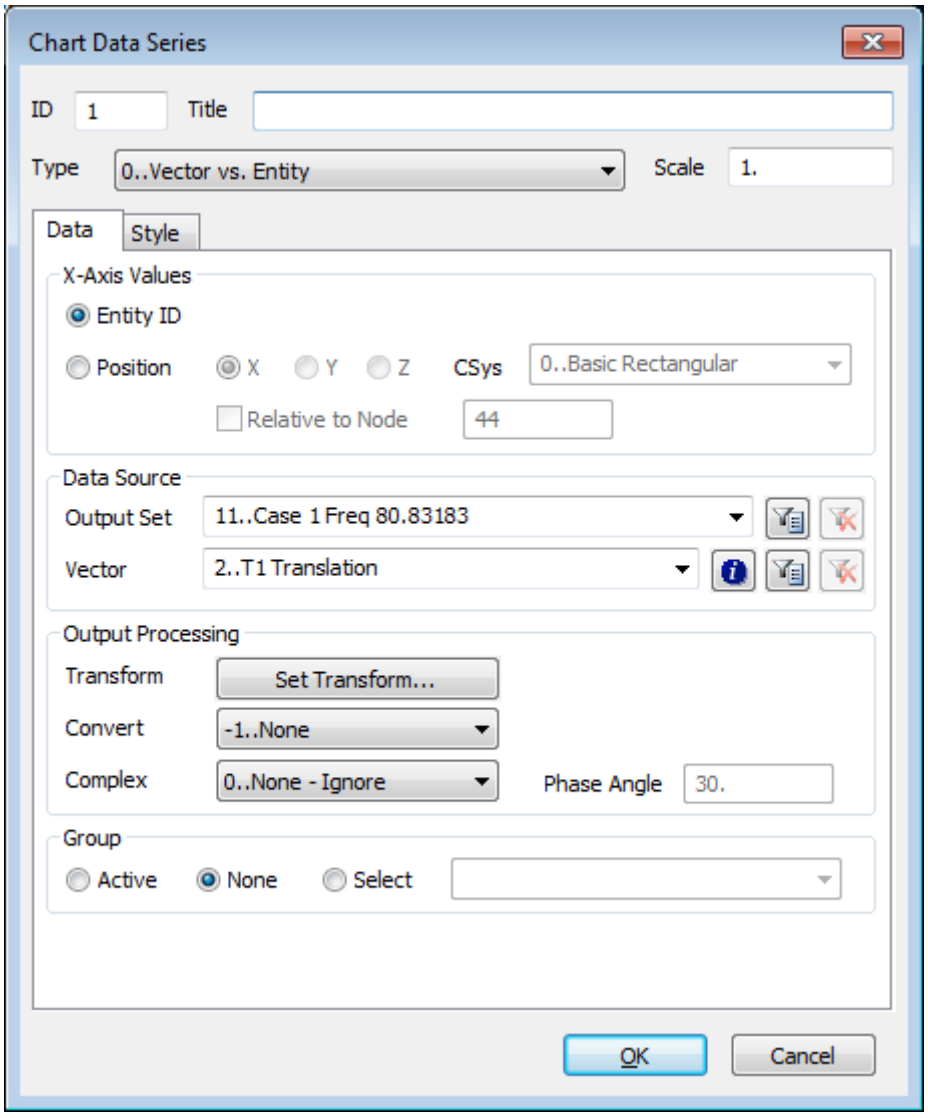
Convert - convert nodal output to elemental output, and vice versa. This option uses the same process as the *Convert* tab of the *Model, Output, Process* command, with "0..Average" and "1..Max Value" corresponding to the options in the *Conversion Approach* section, but does it on-the-fly instead of 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*.



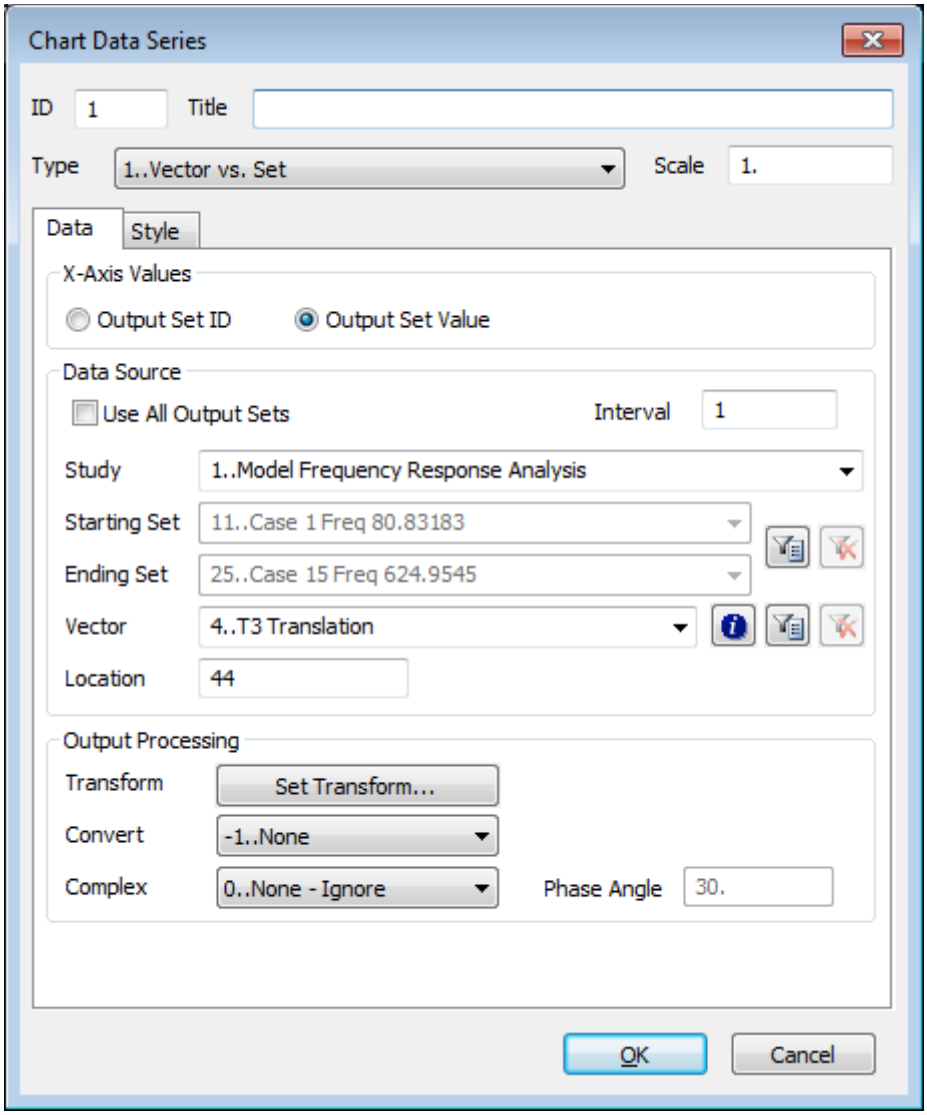
The *X-Axis Values* section offers two options, *Entity ID* or *Position*. When *Entity ID* is selected, the node or element ID of each data point will be used as the X value. When *Position* is selected, the *X*, *Y*, or *Z* coordinate of each node or the centroid of each element will be used as the 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

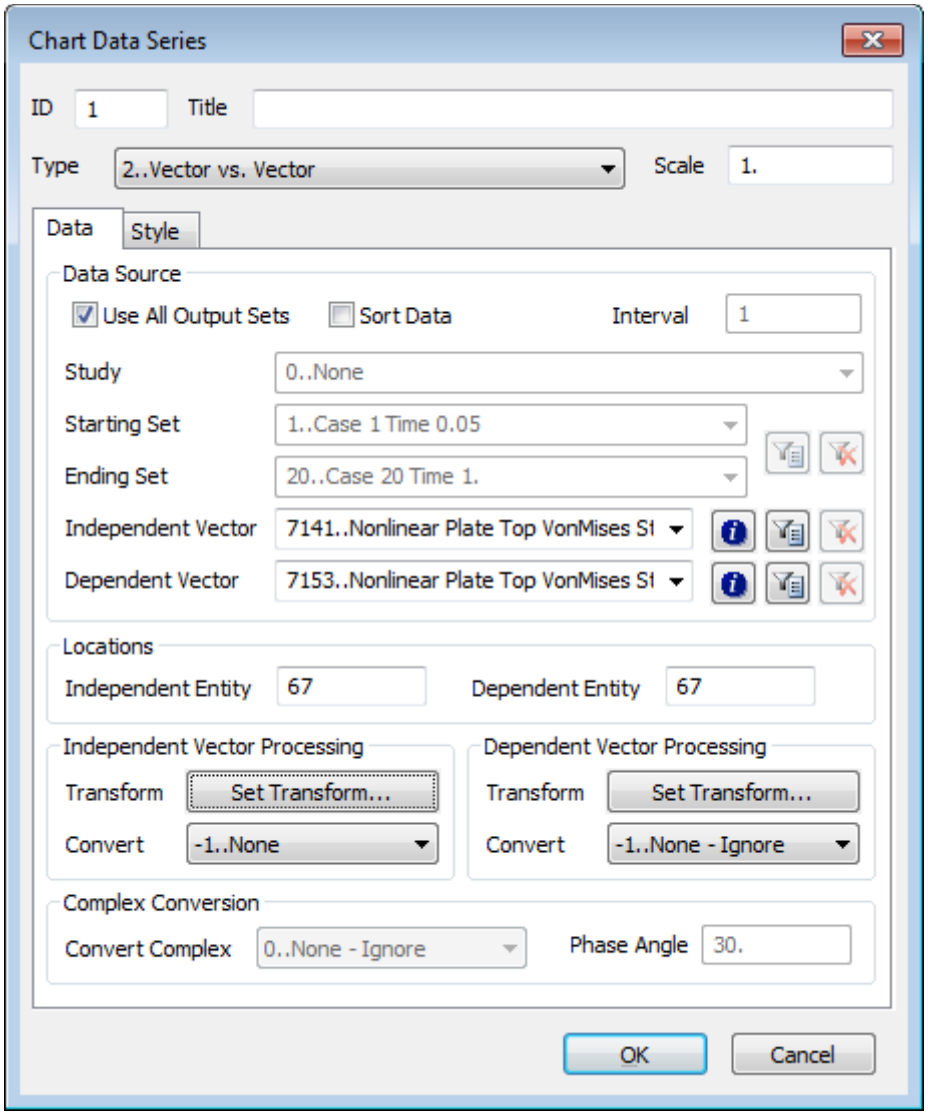
from the *Study* drop-down or by using the *Starting Set* and *Ending Set* drop-downs. Optionally, an *Interval* value can be specified, so only every n-th output set in the range will be used.

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.

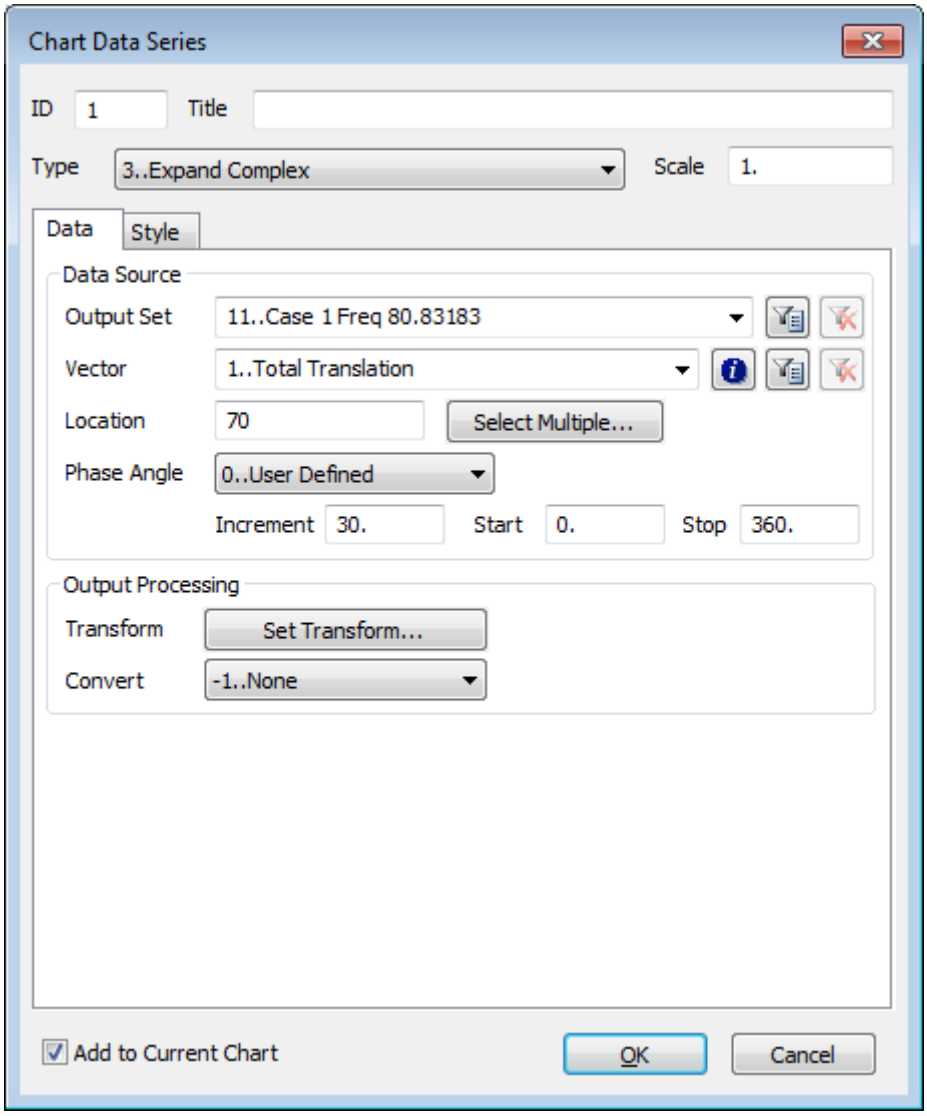
Note: When *Sort Data* is on, lines will be drawn through the data points based on ascending X-values. Also, when listing values or sending them to the clipboard, the X-values will be in ascending order.

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

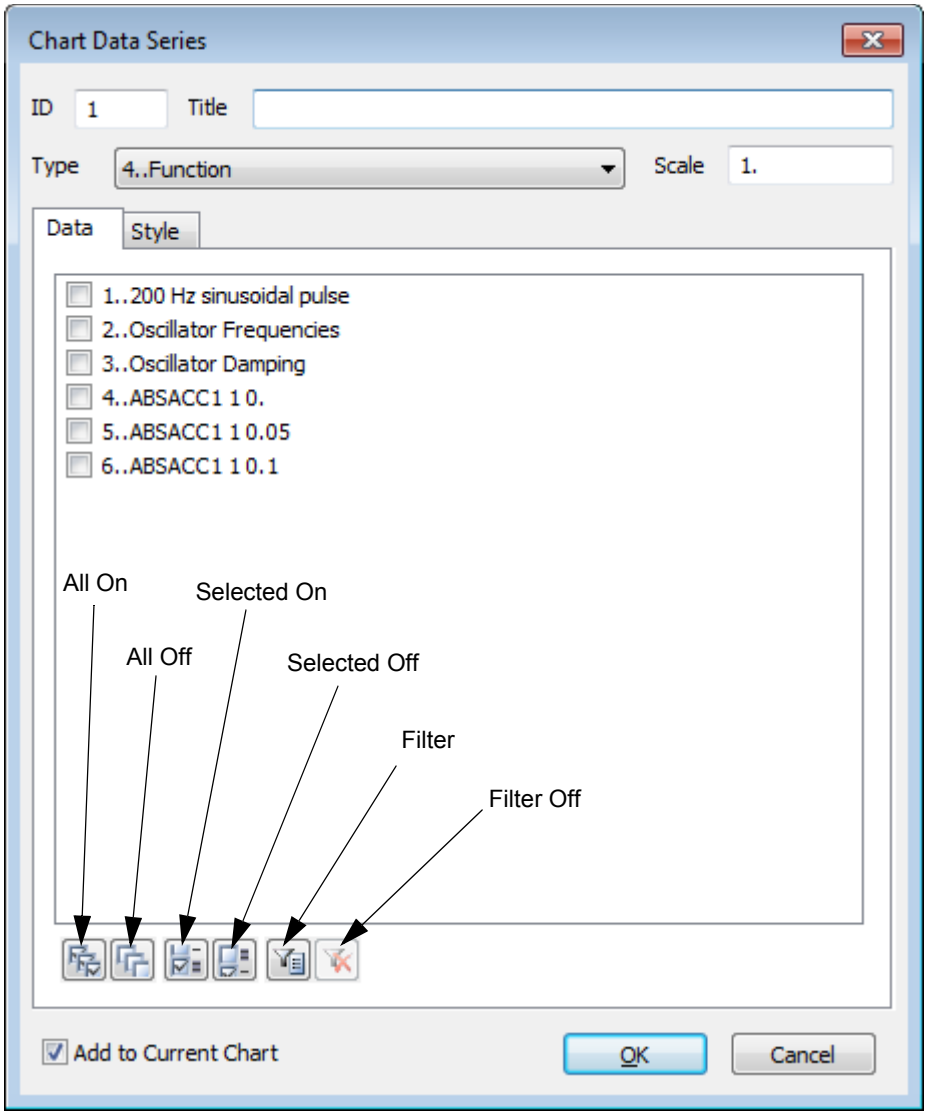
new *Data Series* which use all of the current settings, except for *Location*. The *Location* for each *Data Series* is determined based on selected node or element IDs.

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:

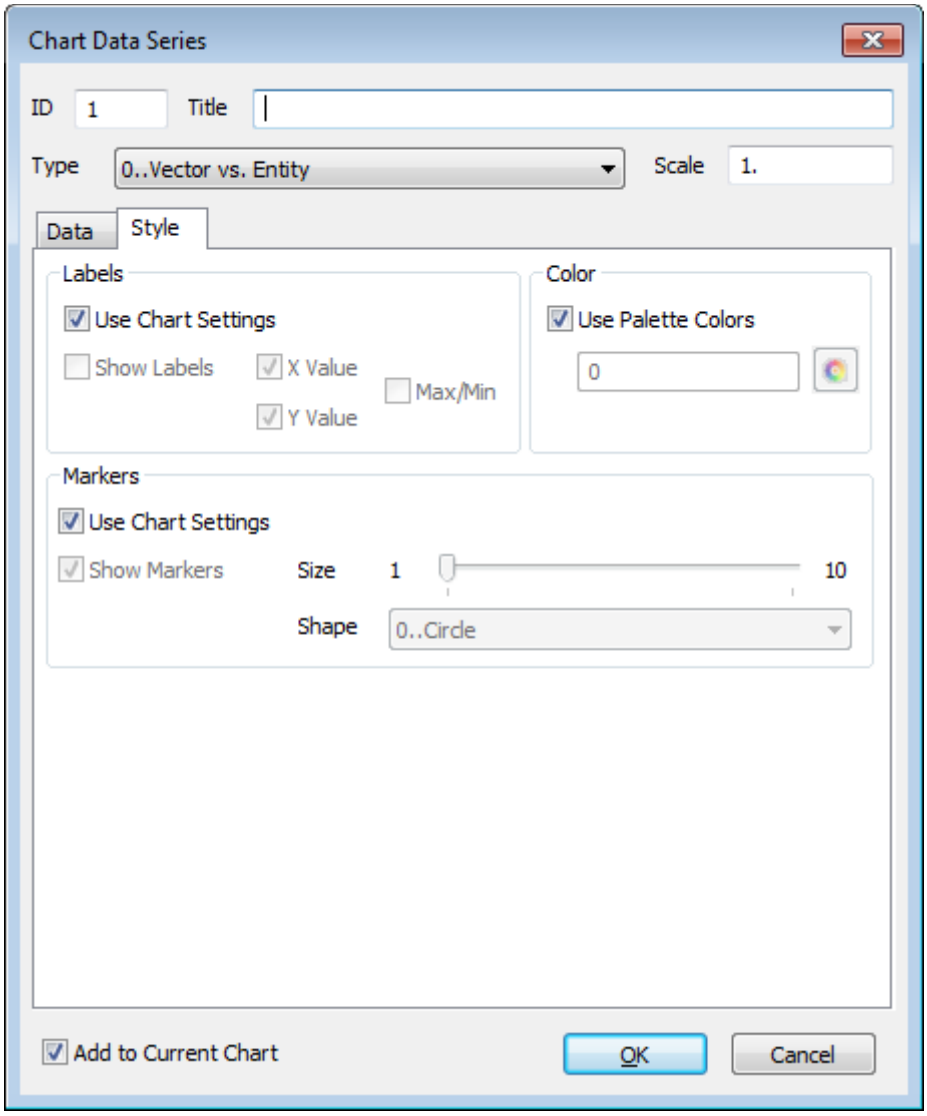


It is also possible to highlight any number of Functions from the list. Holding down the "Ctrl" key will enable highlighting of multiple Functions. Holding down "Shift" while picking a "first" Function, then a "last" Function will highlight multiple. Once highlighting is complete, use the *Toggle Selected 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 Setting*s 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_ElemQualPyrEPIAVal, 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*.