What’s New in FEMAP 20.1

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

VERY IMPORTANT - Output Vector ID Ranges Have Changed!

User Interface

Geometry

Meshing

Elements

Loads and Constraints

Simulation Entities - New for 2020.1!

Connections (Regions, Properties, and Connectors)

Aeroelasticity

Listing

Groups and Layers

Views

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

Tools

Model Merge

OLE/COM API

Preferences

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

Geometry Interfaces

Analysis Program Interfaces

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

User Interface

Geometry

Meshing

Elements

Properties

Layups

Loads and Constraints

Connections (Regions, Properties, and Connectors)

Optimization

Aeroelasticity

Listing

Groups and Layers

Views

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

Tools

Libraries

OLE/COM API

Preferences

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

User Interface

Geometry

Meshing

Elements

Materials

Properties

Layups

Loads and Constraints

Connections (Regions, Properties, and Connectors)

Optimization

Listing

Groups and Layers

Views

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

Tools

Model Merge

OLE/COM API

Preferences

What’s New - version 2020.1

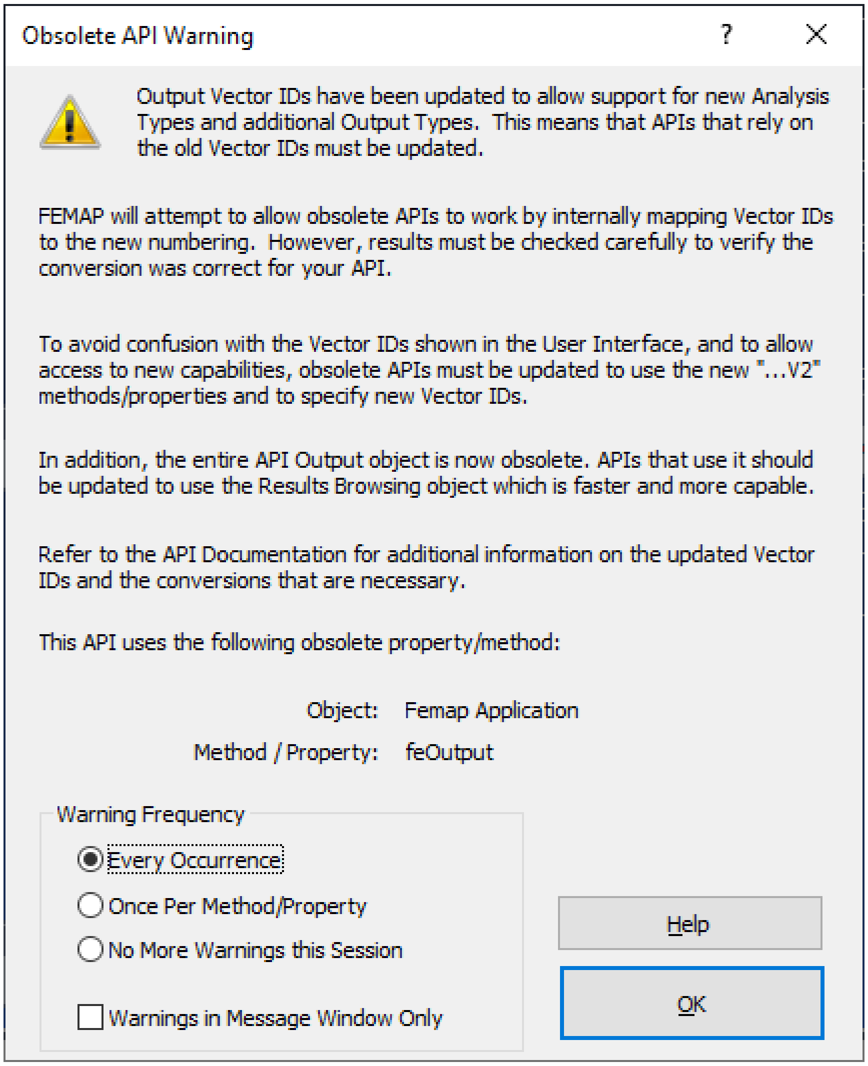
VERY IMPORTANT - Output Vector ID Ranges Have Changed!

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

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

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

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



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

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

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

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

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

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

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

User Interface

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

General

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

Menu

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

Toolbars

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

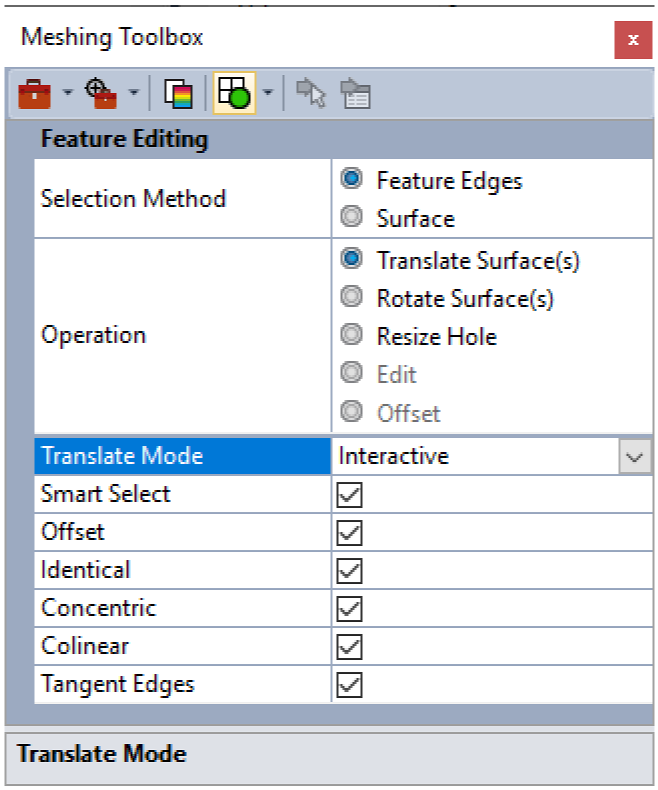
Model Info tree

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

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

Meshing Toolbox

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



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

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

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

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

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

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

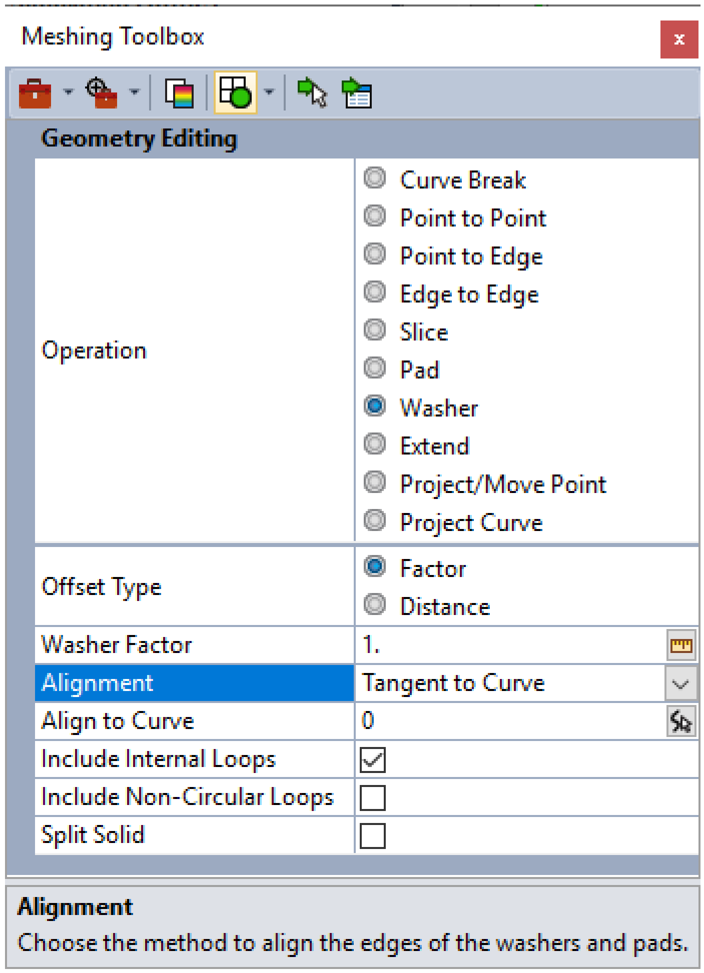
The *Translate Mode* drop-down only appears when *Operation* is set to *Translate Surface(s)*. It has three options:

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

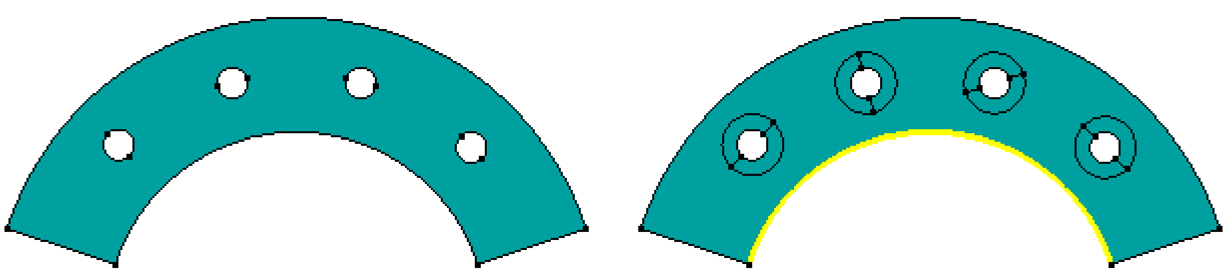
Interactive - First, select any number of surfaces or feature edges, then specify a vector using the standard vector definition dialog box, then click *OK*.

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

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



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



Function/Table Editor

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

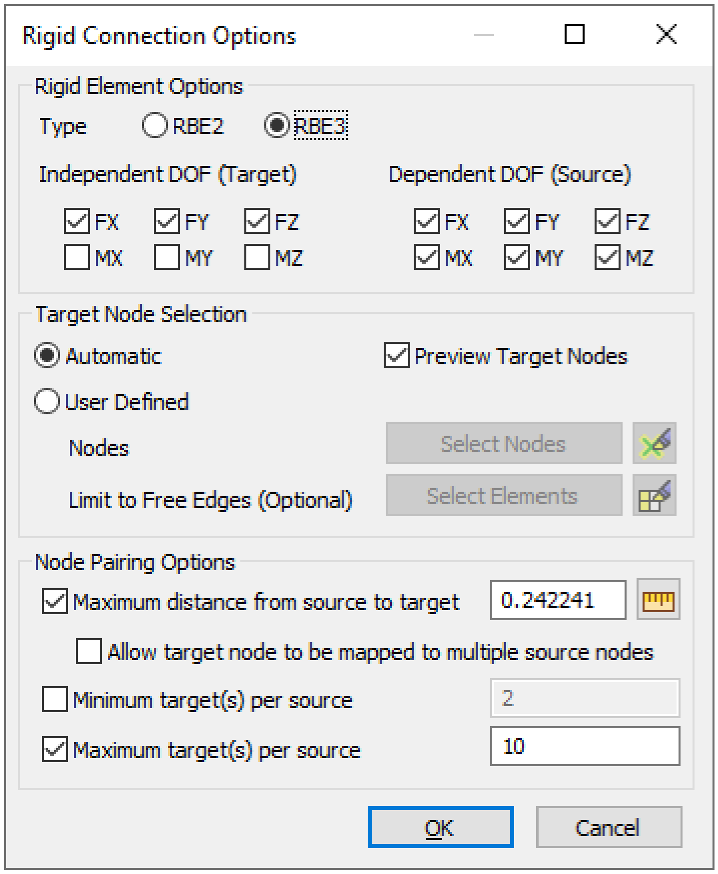
Geometry

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

Meshing

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

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



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

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

Elements

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

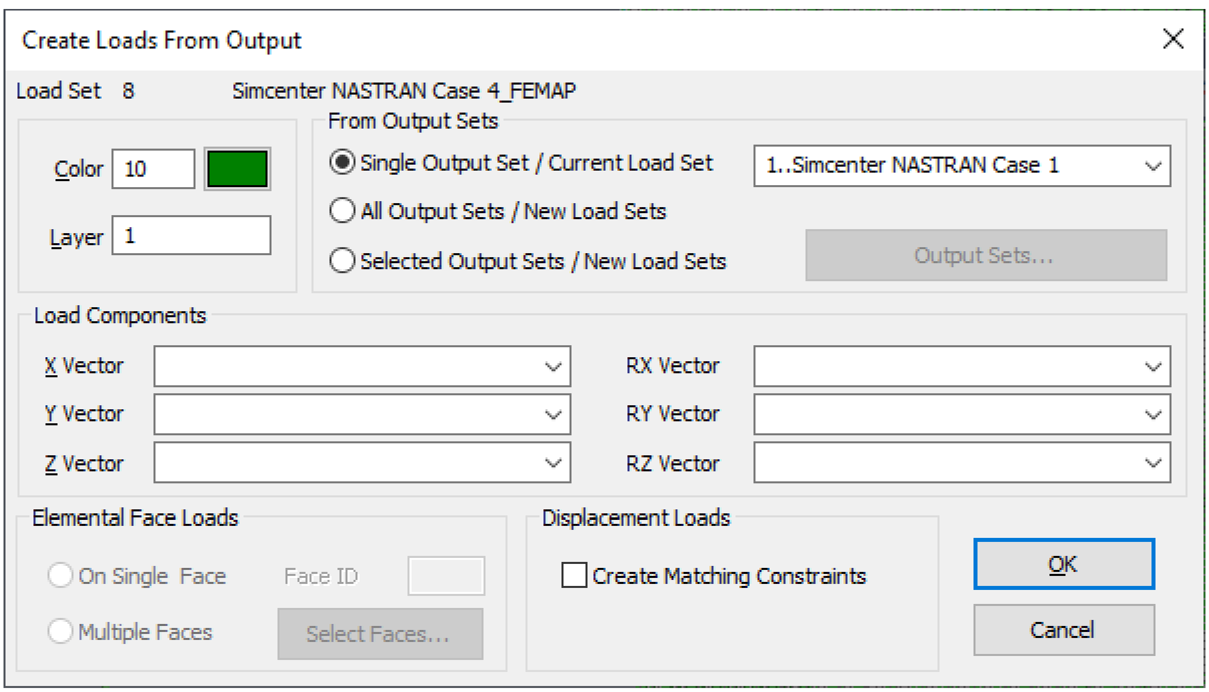
Loads and Constraints

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

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

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

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



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

The *From Output Sets* section offers three options:

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

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

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

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

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

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

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

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

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

Simulation Entities - New for 2020.1!

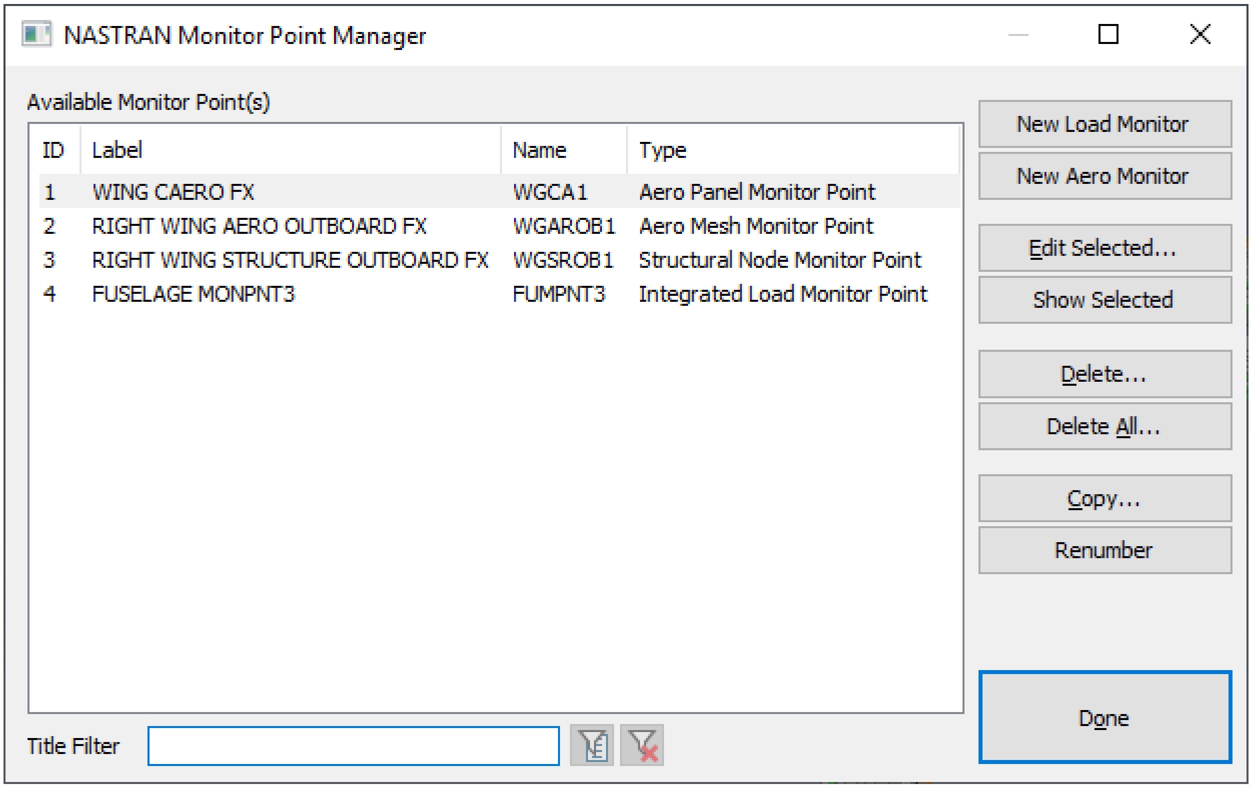
- Added the *Model, Simulation Entities...* menu. The commands on the *Model, Simulation Entities...* menu are used to create, edit, or manage entities, Monitor Points or Direct Matrix Input entities, which can be useful in certain types of analysis performed with Simcenter Nastran or MSC Nastran. These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

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

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

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

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



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

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

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

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

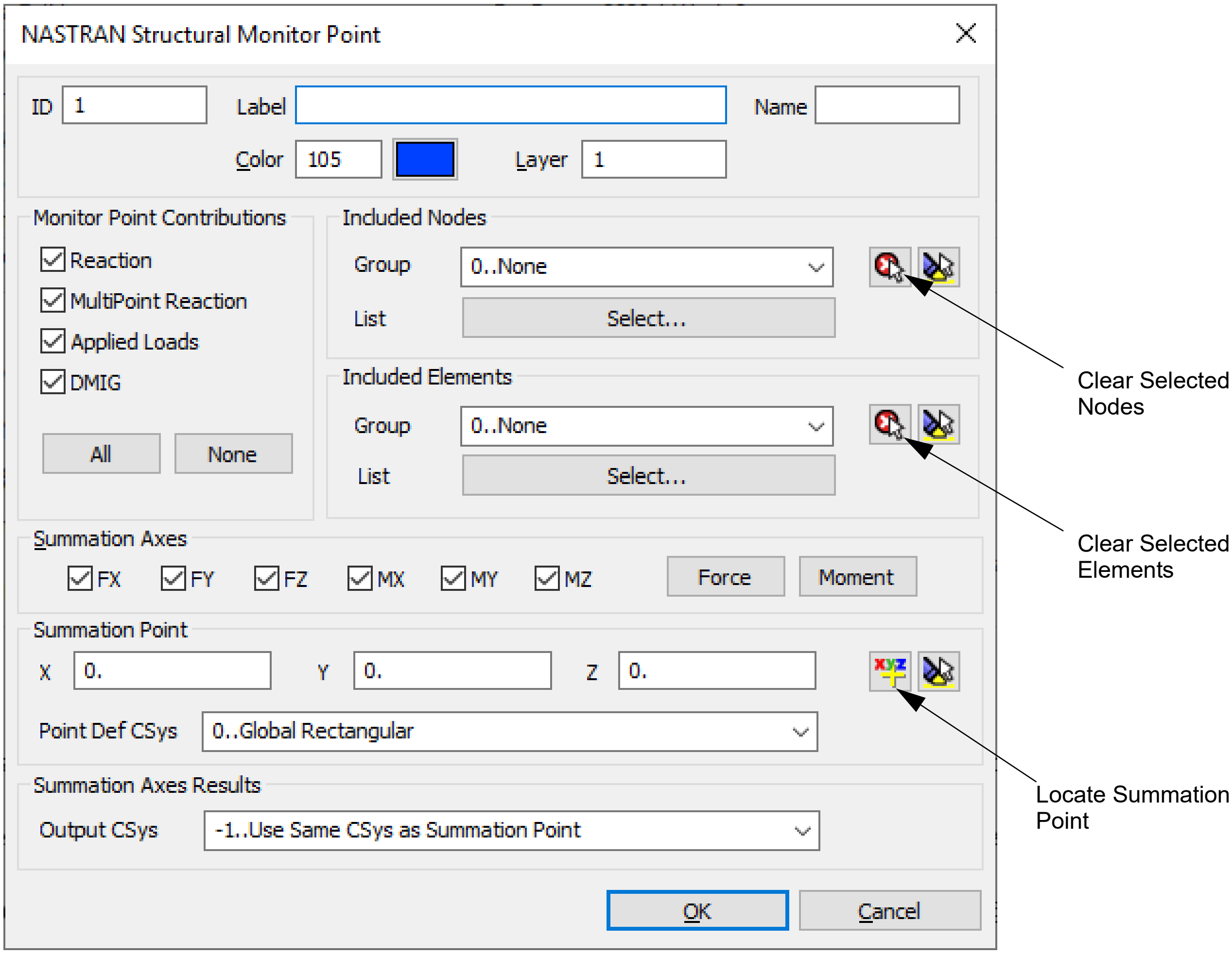
This manager can be used to create a *New Load Monitor* or *New Areo Monitor*; *Edit, Delete*, *Copy*, or *Renumber* the monitor point currently highlighted in the *Available Manufacturing constraint(s)* list; highlight entities in the graphics window which are referenced by the monitor point(s) currently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

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

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

NASTRAN Structural Load Monitor Point

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



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

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

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

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

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

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

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

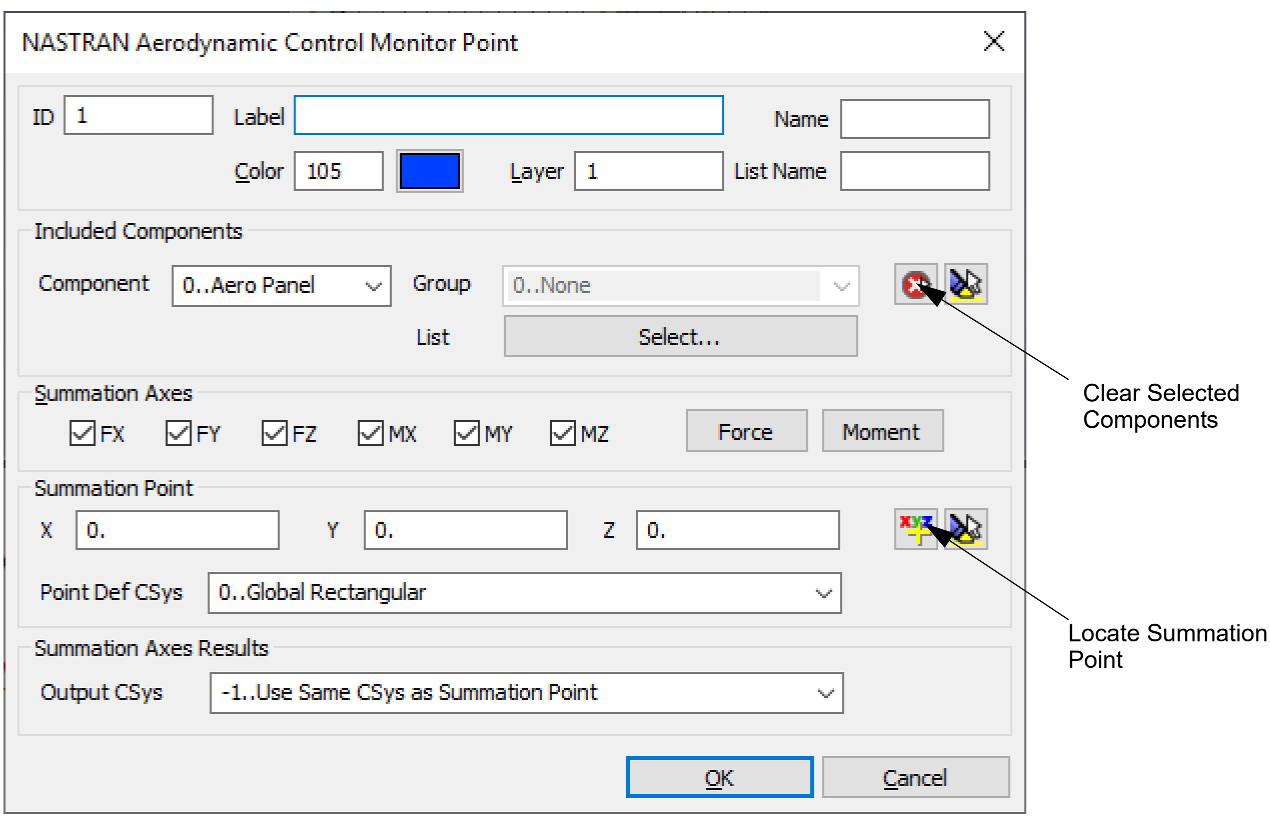
The *Summation Axes* section contains the component axes about which to sum. Any combination of the check boxes in this section being enabled or disabled is allowed, but at least one must be enabled. The *Force* button can be used to quickly toggle the *FX*, *FY*, and *FZ* check boxes on/off, while the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

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

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

NASTRAN Aerodynamic Controls Monitor Point

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



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

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

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

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

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

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

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

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

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

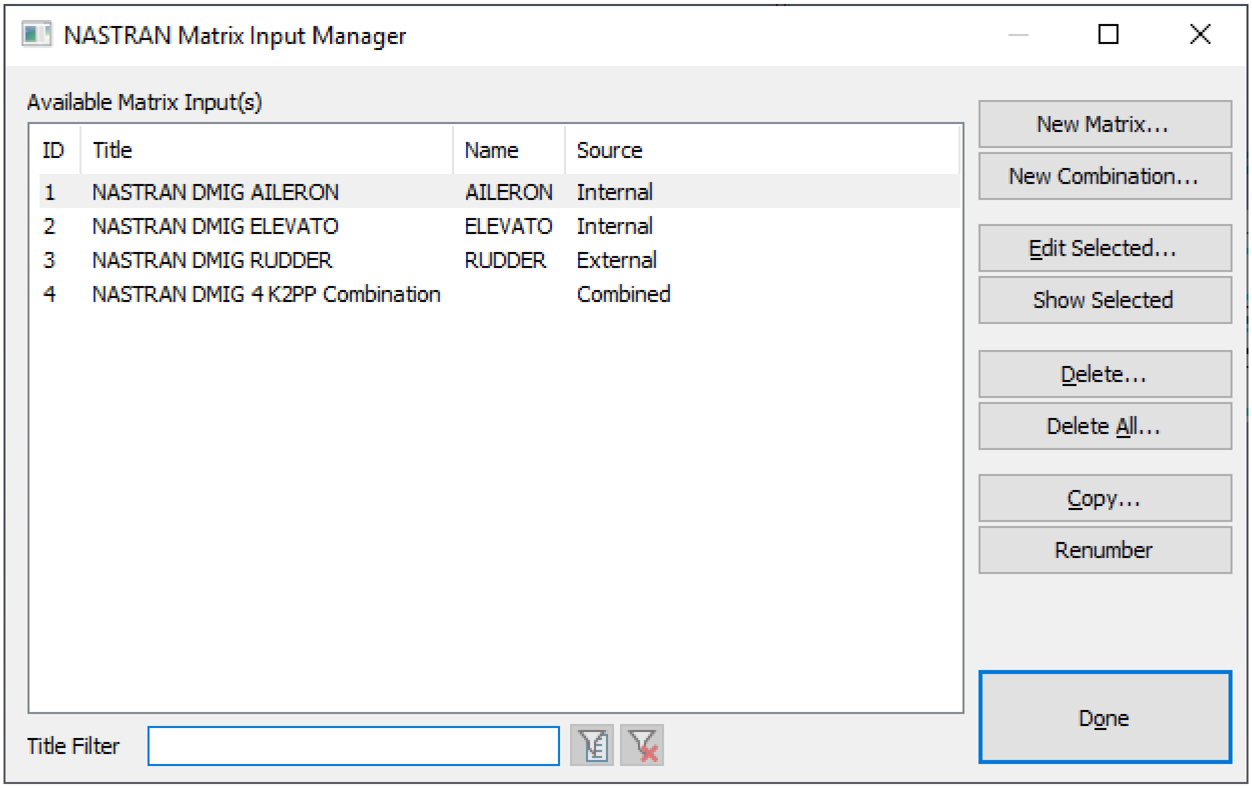
The *Summation Point* section is used to specify the coordinates about which the forces are to be summed. These coordinates are defined in relation to the coordinate system selected via the *Point Def CSys* drop-down. The "Locate Summation Point" icon button can be used to select a location from the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

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

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

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

This command Opens the *NASTRAN Matrix Input Manager*:



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

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

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

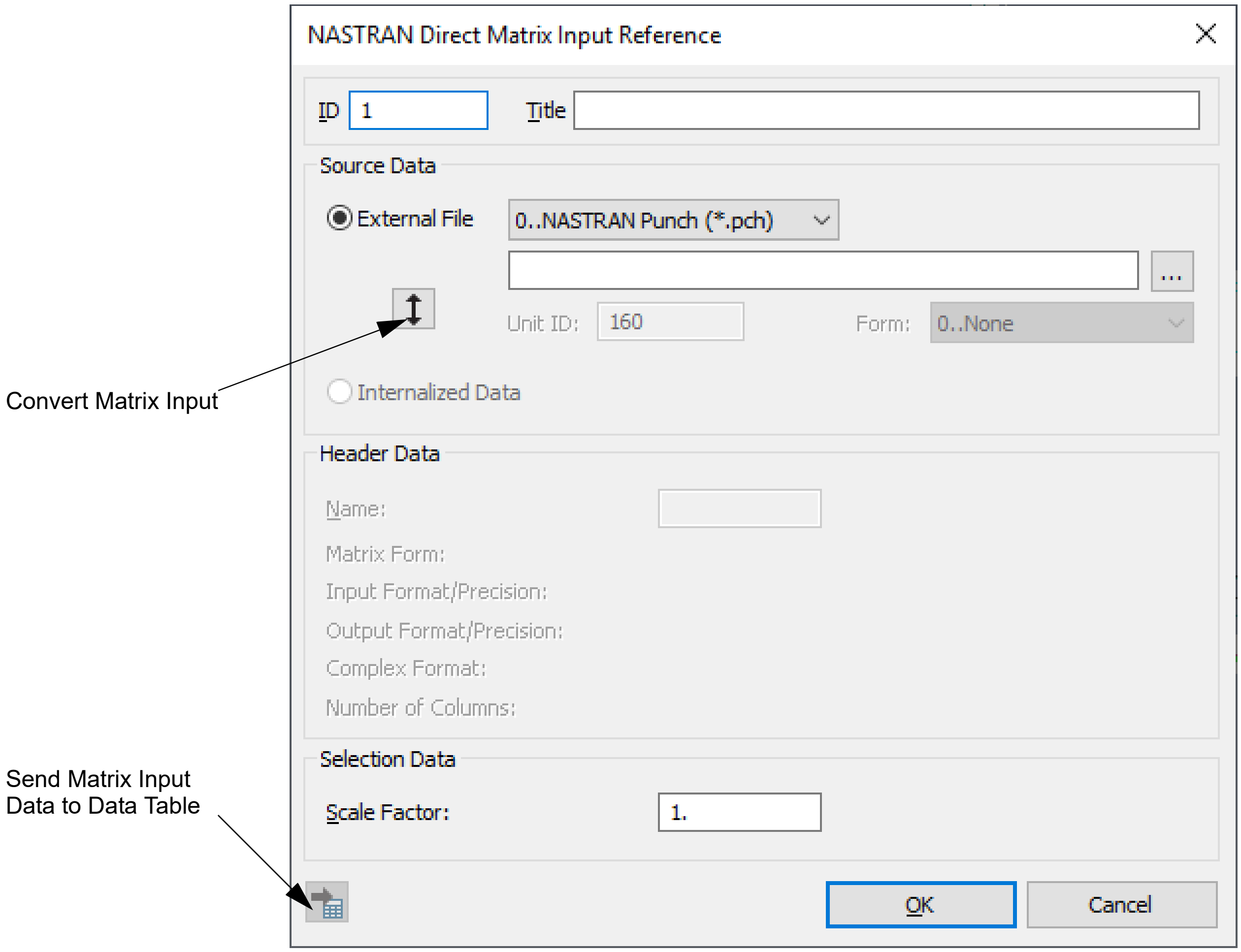
Clicking the *New Matrix* button opens the *NASTRAN Direct Matrix Input Reference* dialog box, which is used to create either an actual DMIG entry in the Nastran input file (*Internalized Data*) or an INCLUDE statement to the Nastran input file which references an external file containing a DMIG entry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

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

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

NASTRAN Direct Matrix Input Reference

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



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

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

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

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

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

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

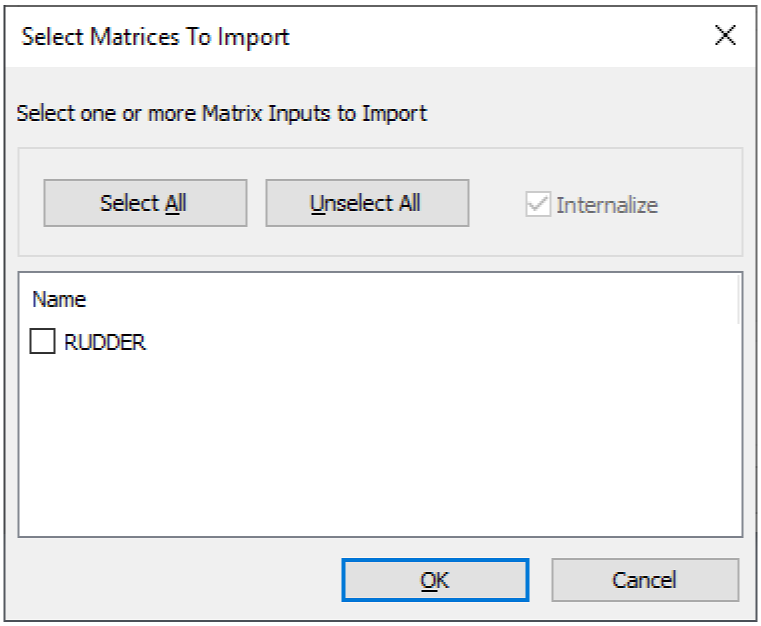
Input Format/Precision - Type of matrix being import. Options are “Real, Single Precision” (Value on DMIG = 1), “Real, Double Precision” (2), “Complex, Single Precision” (3), or “Complex, Double Precision” (4), which are read from/written to the TIN field on the DMIG entry.

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

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

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

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



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

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

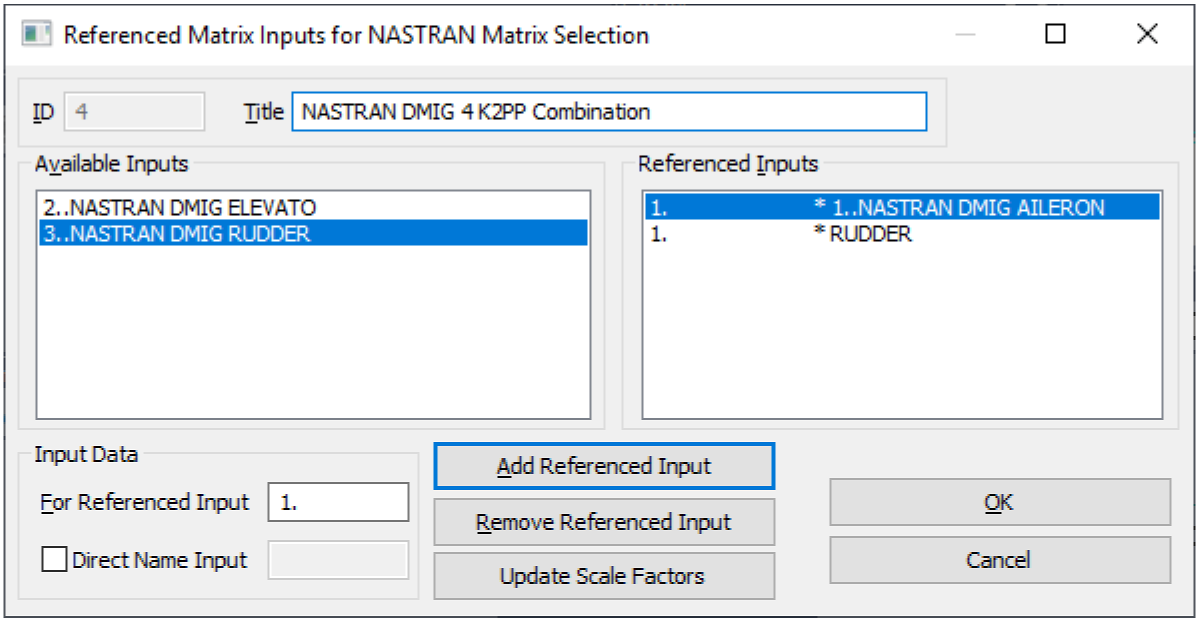
The “Send Matrix Input Data to Data Table” icon button will only be available when *Source Data* is set to *Internalized Data*. Clicking this button will send the direct matrix input data to the *Data Table* in a format similar to this:

Data Table

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

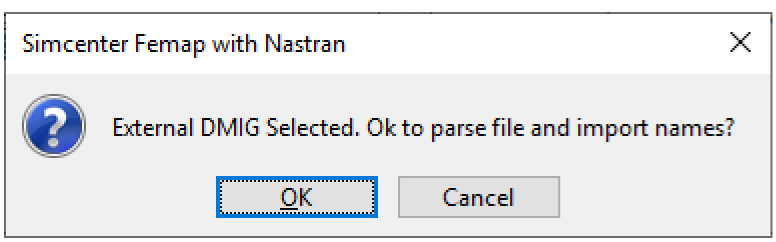
Referenced Matrix Inputs for NASTRAN Matrix Selection

This dialog box is used to create Matrix Combination entries:



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

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



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

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

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

Connections (Regions, Properties, and Connectors)

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

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

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

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

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

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

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

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

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

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

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

“-1..No Regularization” - use no regularization.

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

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

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

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

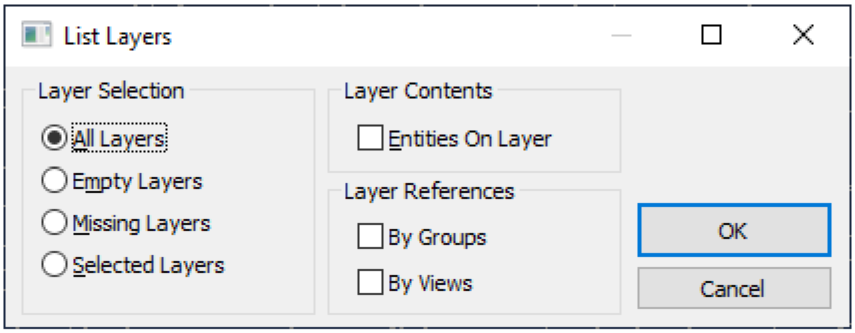
Aeroelasticity

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

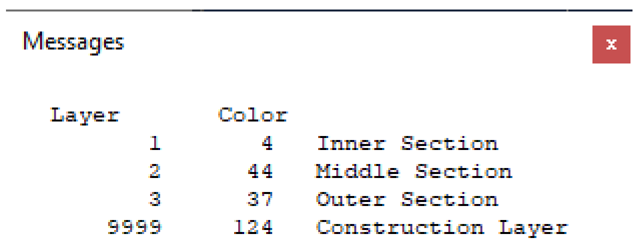
Listing

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

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



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



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

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

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

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

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

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

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

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

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

Messages

List Layers

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

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

Groups and Layers

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

Views

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

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

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

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

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

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

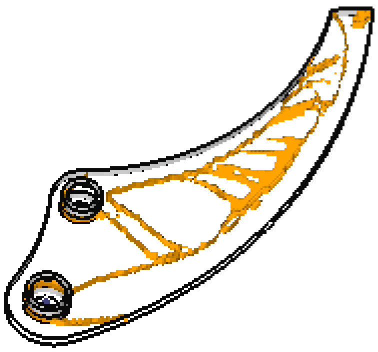
The default setting for digit control for all of the above view options is “Use Preferences”. All of the other available settings are the same as the options available via the *Options...* button in the preferences, so again, see Preferences section for more information on the available settings.

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

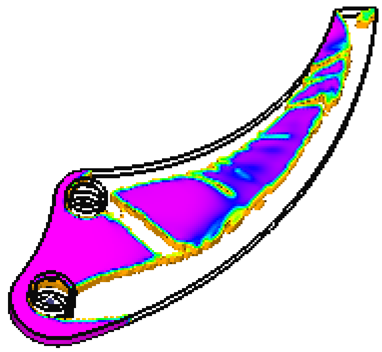
Output and Post-Processing

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

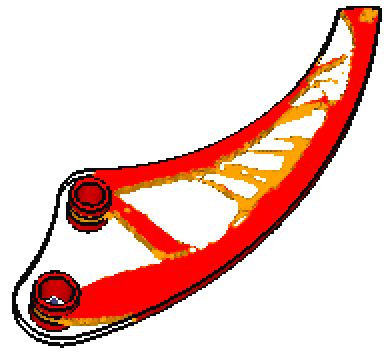
Single IsoSurface at Value



Negative Capped IsoSurface

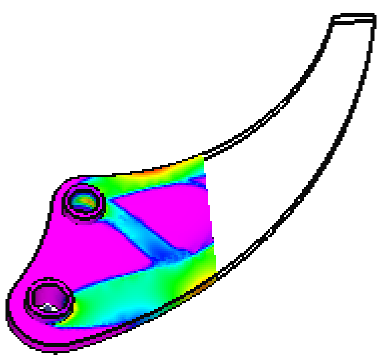


Positive Capped IsoSurface

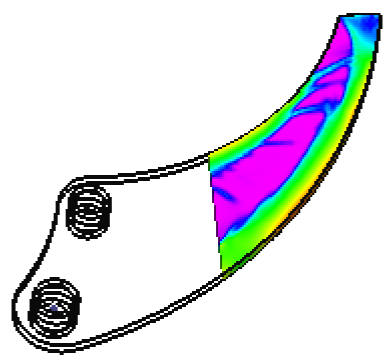


- Added the ability when plotting a *Section Cut,* set to *Cut Model*, to choose if the display is showing all elements on the side of the plane toward the positive plane normal are removed ("0..Negative Cap") or all elements on the side of the plane toward the negative plane normal are removed ("1..Positive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

Section Cut, Negative Cap



Section Cut, Positive Cap



Geometry Interfaces

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

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

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

Analysis Program Interfaces

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

- FEMAP Neutral File Interface
- Simcenter Nastran Interface (Previously NX Nastran)
- Nastran Interfaces (Simcenter and MSC/MD)
- ANSYS Interface
- ABAQUS Interface
- LS-DYNA Interface

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

FEMAP Neutral File Interface

- Updated Neutral Read and Write for v2020.1 changes

Simcenter Nastran Interface (Previously NX Nastran)

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

SOL 401 and SOL 402 Only

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

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

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

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

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

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

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

SOL 401 Only

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

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

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

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

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

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

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

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

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

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

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

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

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

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

"0..Norm 1 Criteria" - Use Norm 1 Criteria. Writes value of L1.

"1..Norm 2 Criteria" - Use Norm 2 Criteria. Writes value of L2.

"2..Norm Infinity Criteria" - Use Norm Infinity Criteria. Writes value of LINF.

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

"0..Norm 1 Criteria" - Use Norm 1 Criteria. Writes value of L1.

"1..Norm 2 Criteria" - Use Norm 2 Criteria. Writes value of L2.

"2..Norm Infinity Criteria" - Use Norm Infinity Criteria. Writes value of LINF.

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

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

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

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

SOL 402 Only

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

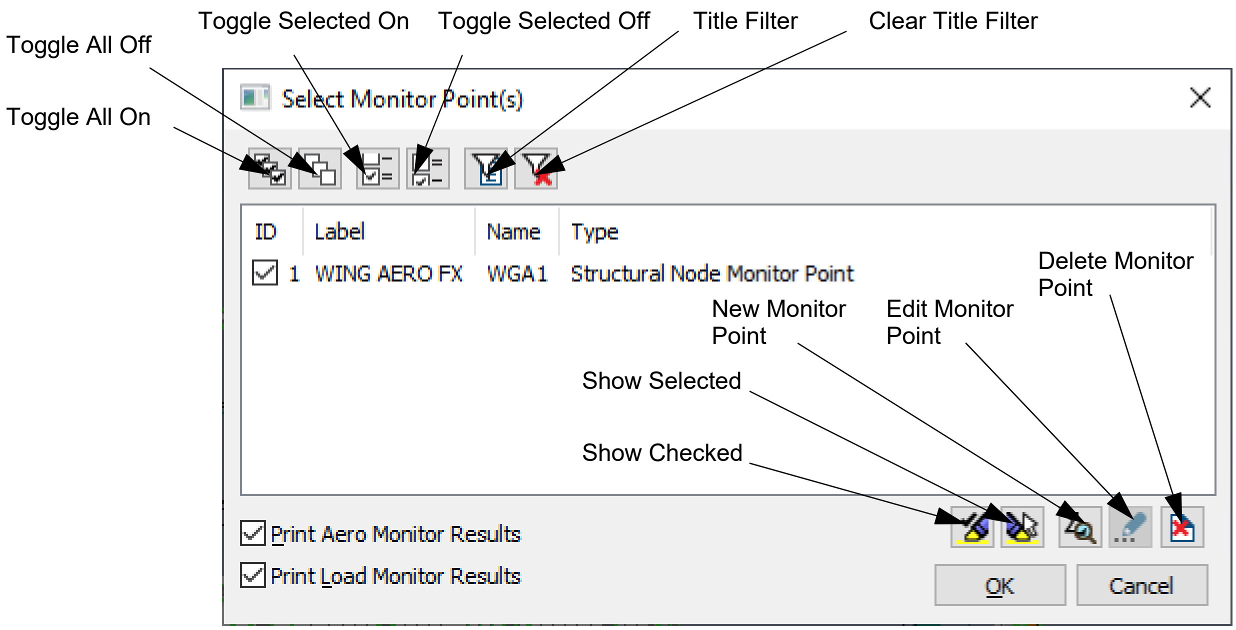
"2..Enable based on PRCE, PRCQ, PRCR, and REFP" - Writes value of 2.

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

Nastran Interfaces (Simcenter and MSC/MD)

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

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



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

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

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

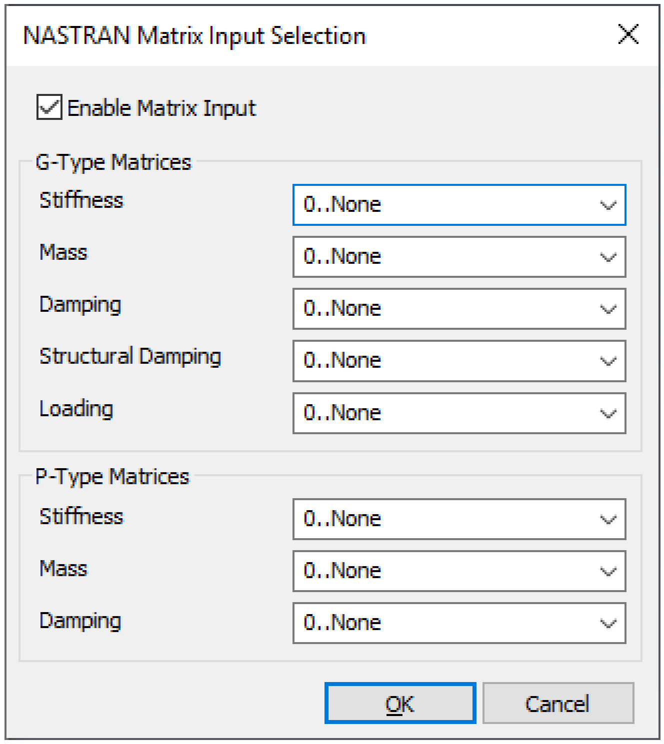
The *Show Checked* and *Show Selected* icon buttons are used to highlight the included entities for each monitor point in the graphics window. Using *Show Checked* will highlight the included entities for each monitor point checked in the *Select Monitor Point(s)* dialog box, while *Show Selected* uses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

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

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

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

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



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

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

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

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

Stiffness - Writes K2GG Case Control entry

Mass - Writes M2GG Case Control entry

Damping - Writes K2GG Case Control entry

Structural Damping - Writes K42GG Case Control entry

Loading - Writes P2G Case Control entry

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

Stiffness - Writes K2PP Case Control entry

Mass - Writes M2PP Case Control entry

Damping - Writes B2PP Case Control entry

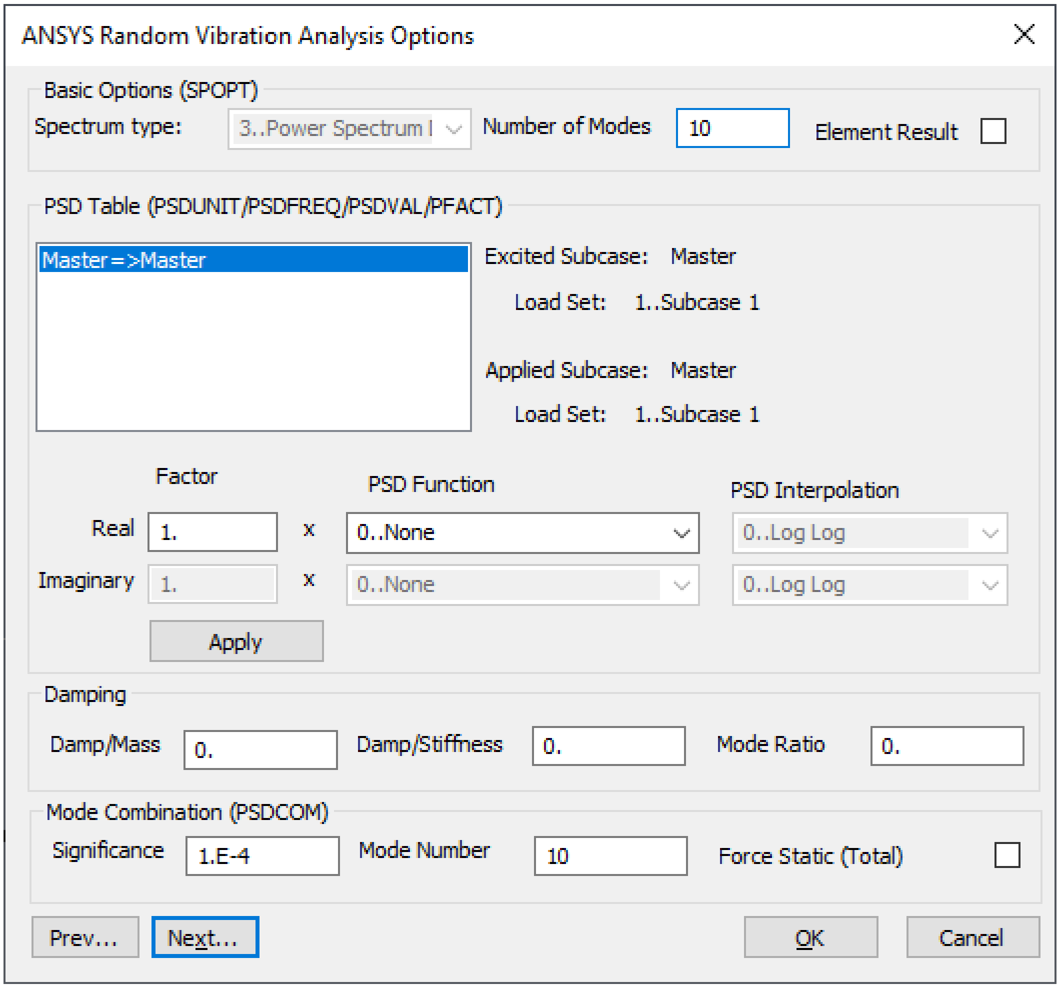
A number of bugs were corrected

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

ANSYS Interface

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

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



Basic Options (SPOPT)

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

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

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

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

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

PSD Table (PSDUNIT/PSDFREQ/PSDVAL/PFACT)

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

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

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

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

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

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

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

Damping

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

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

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

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

Mode Combination (PSDCOM)

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

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

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

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

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

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

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

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

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

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

A number of bugs were corrected.

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

ABAQUS Interface

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

A number of bugs were corrected.

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

LS-DYNA Interface

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

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

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

A number of bugs were corrected.

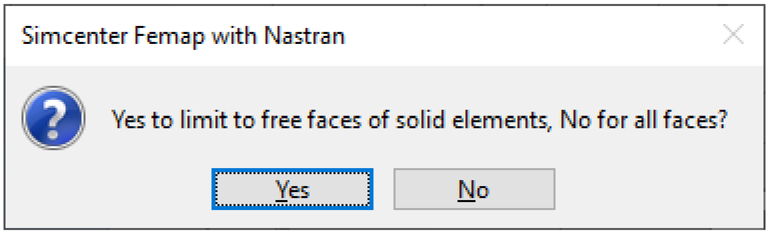
For details, see Section 8, “Analysis Program Interfaces” in the *FEMAP User Guide*.

Tools

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

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

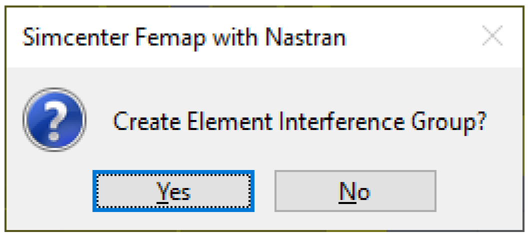
You first select the elements to be checked, then answer up to two additional questions. If any solid elements are selected, the following question will appear first:



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

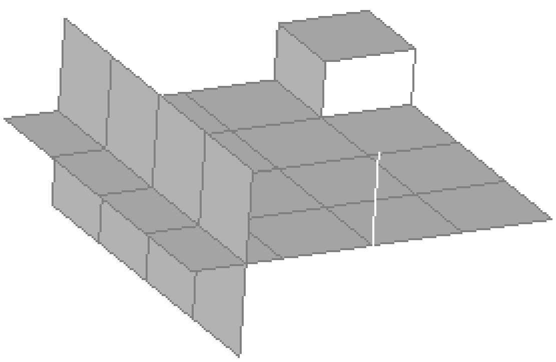
Any elements which interfere with each other will be highlighted in the graphics window using the current settings for the *Window, Show Entities* command.

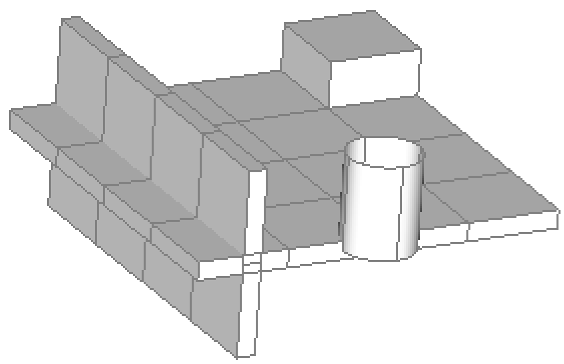
In all cases where elements are found to be interfering with one another, the following question will appear:



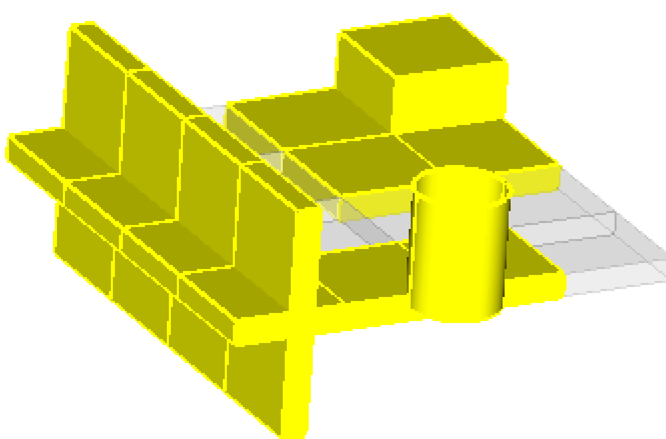
Select *Yes* to automatically create a group with all elements which interfere with at least one other element.

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





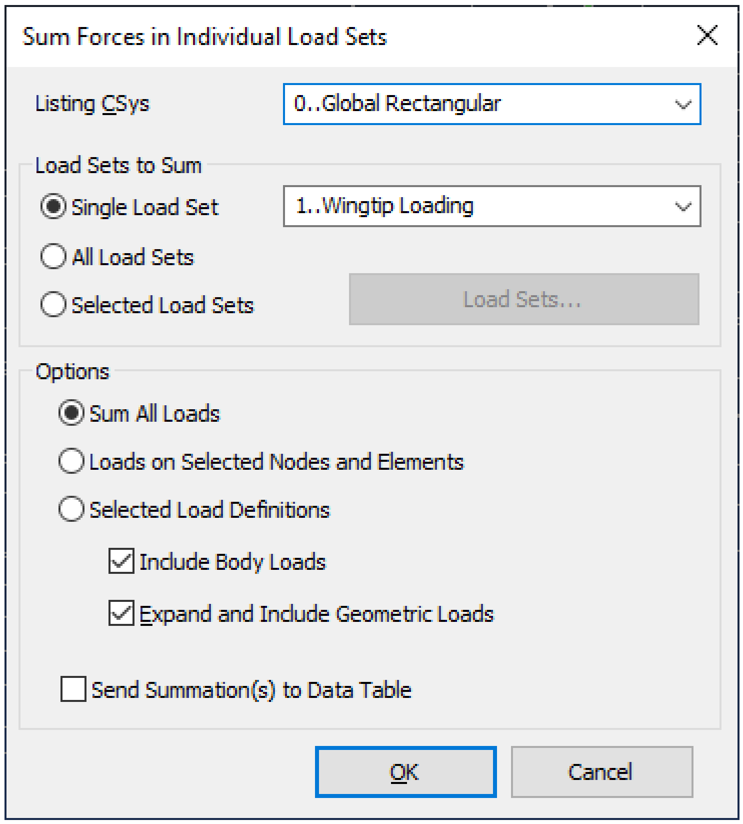
Would find that the highlighted elements interfere with at least one other element:



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

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

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



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

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

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

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

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

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

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

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

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

Messages

Check Sum of Forces

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

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

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

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

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

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

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

Model Merge

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

OLE/COM API

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

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

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

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

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

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

- Removed Output Object and all related methods.

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

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

- feDeleteOutputEntryV2
- feSelectOutputV2
- feSelectOutput2V2

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

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

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

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

New and modified API Objects and Attributes

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

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

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

New and Updated API Methods

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

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

The following functions have been added or updated:

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

New and updated Global Variables

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

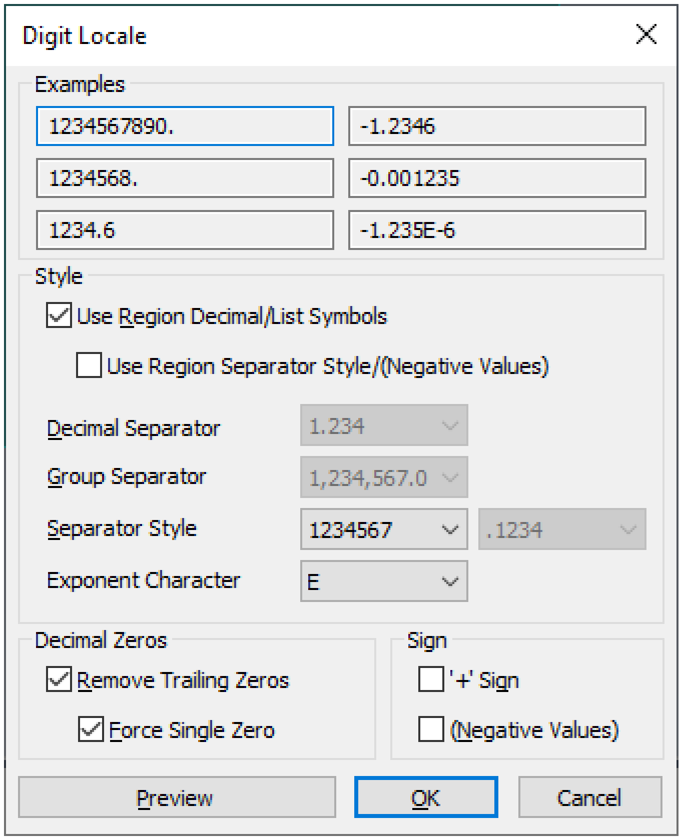
Preferences

Graphics

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

Locale button

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



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

Examples

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

Style

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

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

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

The individual controls in the *Style* section control:

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

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

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

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

Decimal Zeros

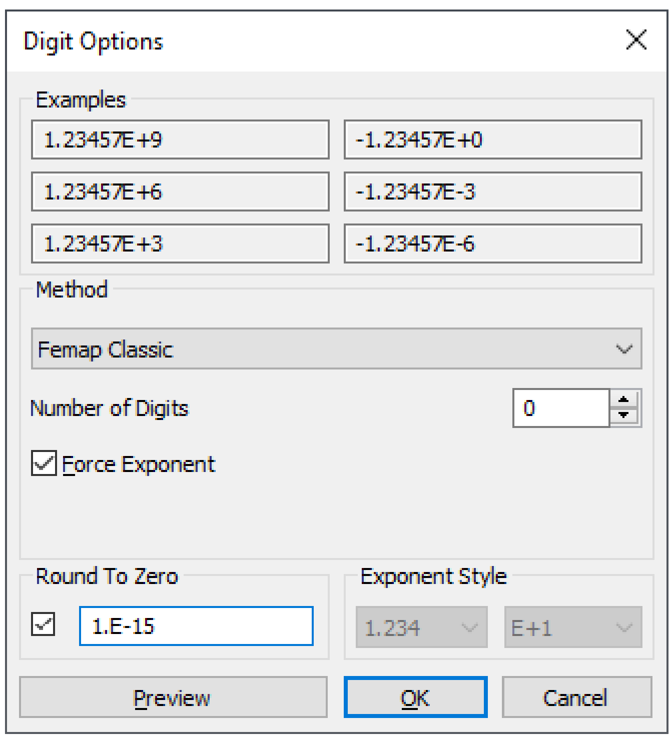
This section controls how zeros are displayed to the right of the *Decimal Separator*. Both options are enabled by default. When *Remove Trailing Zeros* is enabled, any value of 0 which ends a value will be removed (i.e., value of 123.45600 becomes 123.456). When *Force Single Zero* is enabled, a single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

Sign

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

Options button

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



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

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

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

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

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

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

Examples

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

Method

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

Round to Zero

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

Exponent Style

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

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

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

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

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

Interfaces

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