

# *What's New in FEMAP*

# 2022

FEMAP 2020.2 included enhancements and new features, which are detailed below:

User Interface

Geometry

Meshing

Elements

Properties

Layups

Loads and Constraints

Connections (Regions, Properties, and Connectors)

Functions

Aeroelasticity

Listing

Views

Output and Post-Processing

Geometry Interfaces

Analysis Program Interfaces

Tools

Model Merge

OLE/COM API

Preferences

FEMAP 2020.1 MP 2 included enhancements and new features, which are detailed below:

Output and Post-Processing

Analysis Program Interfaces

FEMAP 2020.1 MP 1 included enhancements and new features, which are detailed below:

User Interface

Analysis Program Interfaces

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 2020.2

## User Interface

General, Menu, Toolbars, Meshing Toolbox, Charting, Data Surface Editor, Function/Table Editor, Mesh Point Editor, Connection Editor, Entity Info, Data Table, API Programming

### General

- Added ability to use commands on “floating” toolbars (i.e., not “docked”) while using another command. It was already possible to use commands on “docked” toolbars in previous versions.
- Added “Group Display Mode” (i.e., *Show Full Model*, *Show Active Group*, and *Show Multiple Groups*) commands to the *Commands* tab of the *Customize* dialog box. They are found under “Additional Commands”. Once added to any toolbar, shortcut keys can also be assigned to these commands.
- Added *Send to Excel* command to *Pick*<sup>^</sup> menu for the standard selection dialog box, which automatically opens Microsoft Excel and sends the IDs currently selected to Excel using the same format as *Copy as List*.
- Added ability to keep registry and initialization settings separate for multiple installations on the same system when they use different licensing and/or language versions. Currently licensing using Demo, Subscription, API-only, and Dongle/Network are considered separate, as are English and other language installations.
- Enhanced *Copy to Clipboard* and *Save to File* commands for the *Data Table*, *Connection Editor*, *Mesh Point Editor*, *Function/Table Editor* and *Entity Editor* to copy/save the data using formatted HTML.
- Improved performance and accuracy of the *Along Curve*, *Length Along*, and *Midpoint* options on the *Methods*<sup>^</sup> menu of the standard coordinate definition dialog box.
- Updated *Equation Editor* dialog box to be resizable, added context-sensitive help by pressing the F1 key, and reordered the items in the *Functions* list to group similar items together. In addition, added ELTHK( elemID ; cornerID ), which returns planar element corner thicknesses.
- Updated the cursor to show when FEMAP is busy but the current activity/command can be aborted, using the cursor selected with the current windows themes. Also, enhanced performance when aborting commands used to calculate mass properties, calculate solid properties, measure, check, and list various entities by only checking for abort after a certain number of passes through a loop in the command instead of every single time through the loop. In some cases, this leads to over a 10X performance increase in responsiveness to abort a command. Now uses the cursor selected with the current Windows theme rather than a custom one.
- Updated minimize, maximize, and close icons in the title bar of an undocked graphics window to more closely match the “flat style” of the same icons found in title bar of the main FEMAP application.
- Updated selection of solids to allow use of the *Select Visible* icon, but only selects visible solids, not volumes.

### Menu

- Added *List, Output, Results to Excel* command. See Output and Post-Processing for more information.
- Added *View, Advanced Post, Stress Linearization* command. See Output and Post-Processing for more information.

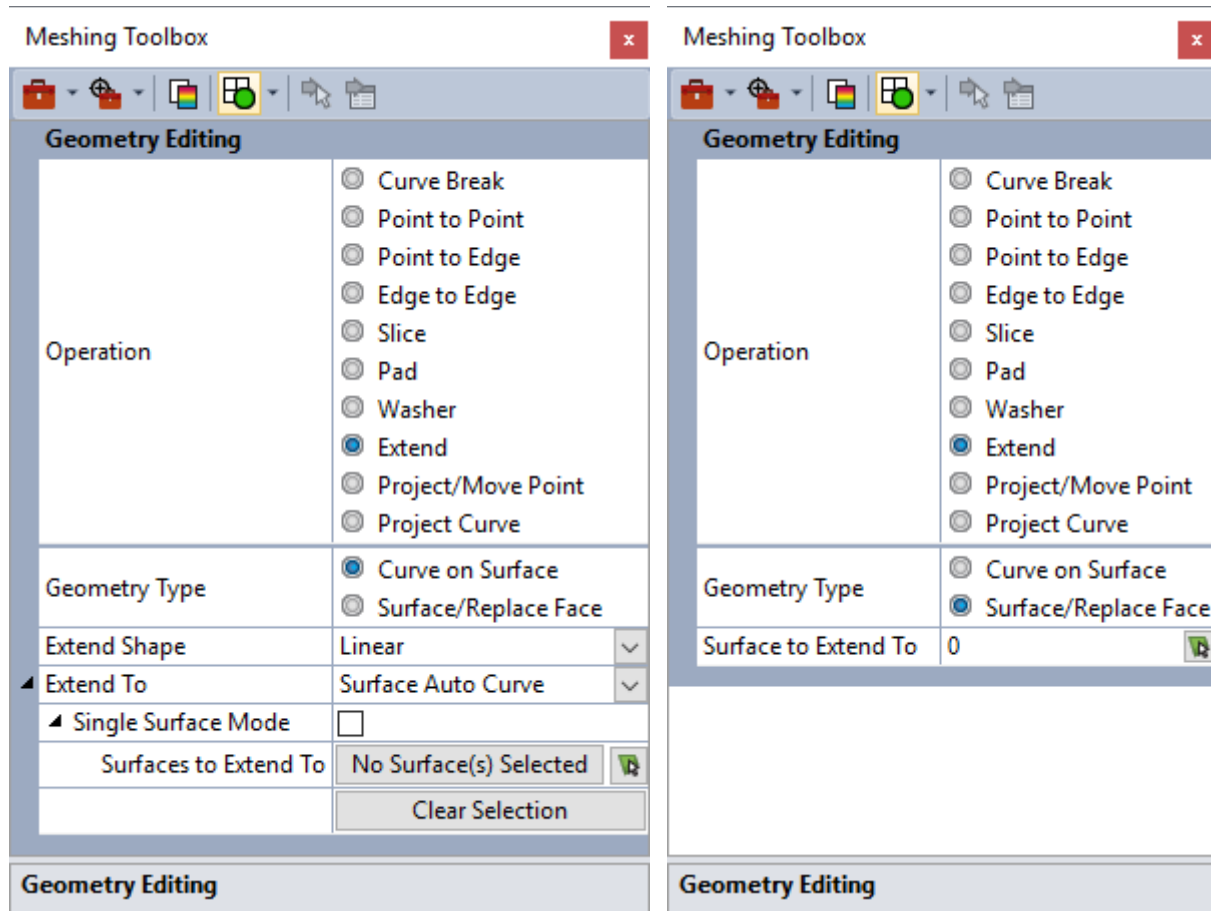
### Toolbars

- Added *Curve Connectivity* to the *View Style* icon menu on the *View - Simple* and *View* Toolbars
- Added curve length and arc/circle info to Tooltips when *Curve* is the active entity in the *Select* Toolbar and Show Tooltips is enabled.

## Meshing Toolbox

- *Feature Removal* tool - Added *Preview* option when *Feature Type* is set to *Blends*. When enabled, the standard entity selection dialog box will appear pre-populated with the blends (i.e., surfaces) to be removed. Simply add or remove surfaces using the standard entity selection dialog box, then click *OK* to proceed with blend removal.
- *Geometry Editing* tool - Added *Geometry Type* option when *Operation* is set to *Extend*.

When *Geometry Type* is set to *Curve on Surface*, mimics the capabilities of the *Geometry*, *Midsurface*, *Extend* command. To use, simply choose a single edge of a surface to extend with the *Entity Select* icon in the *Meshing Toolbox* or multiple edges using *Dialog Select* icon. *Surface* and *Surface Auto Curve* are very similar, with the only difference being that *Surface Auto Curve* will automatically select any number of additional curves to extend, based on the selected curve. For instance, if you choose a curve on the end of a pipe, it will select any additional curve(s) needed to extend the entire pipe. When using *Surface*, only curves you explicitly select will be extended.



Both *Surface* and *Surface Auto Curve* offer two modes for selecting surfaces, controlled by *Single Surface Mode*.

When the *Single Surface Mode* is enabled, toggle on the *Entity Select* icon and you will be prompted to select a surface, then select a curve. Depending on which *Extend To* option you are using, that curve or curves will be extended to only that surface, then you will be prompted to select a surface and curve again.

When *Single Surface Mode* is disabled, any number of surfaces on different solids, surfaces, or general bodies may be selected by clicking the “...” button located next to the “No Surface(s) Selected” button. Once selected, the number of selected surfaces will appear in the text of the button, which when clicked will highlight the selected surfaces. After surface selection is completed, choose any number of curves found on surfaces and they will be extended to the selected surface(s). At any time, use the *Clear Selection* button to reset surface selection.

When *Geometry Type* is set to *Surface/Replace Face* mimics the *Geometry*, *Solid*, *Extend/Replace Face* command. Simply click icon button next to the *Surface to Extend To* field, select a surface and click *OK*, then use *Entity Select* or *Dialog Select* icons in the *Meshing Toolbox* to select surfaces to extend solid(s) to the surface specified for *Surface to Extend To*.

## Charting

- Added *Send to Excel* command to the *Copy to Clipboard* icon menu, which opens Microsoft Excel, then sends both the image and values from the active Chart to Excel.

## Data Surface Editor

- Added *Send Data Surface to Excel* command to the toolbar, which opens Microsoft Excel and transfers text from the *Data Surface Editor* directly to Excel.

## Function/Table Editor

- Added *Send to Excel* command to the *Copy Text to Clipboard* icon menu, which opens Microsoft Excel, then sends all text, including header information (*ID*, *Title*, *Type*, *Column Titles*, and *Row IDs*) directly to Excel.

## Mesh Point Editor

- Added *Send to Excel* command to the *Create/Manage Mesh Points* icon menu, which opens Microsoft Excel and sends all Mesh Point data directly to Excel in a specific format, via the clipboard. This data can then be pasted back into the same FEMAP model or a different model as long as the data remains on the clipboard in the format created by the command.

## Connection Editor

- Added *Send to Excel* command to the toolbar, which opens Microsoft Excel and sends the current data in the *Connection Editor* directly to Excel.

## Entity Info

- Added curve length and arc/circle info to when *Curve* is the active entity in the *Select* Toolbar.

## Data Table

- Added *Send to Excel* command to the toolbar, which opens Microsoft Excel and sends the current data in the *Data Table* directly to Excel.
- Updated *Data Table* to be unlocked by default when it is opened.

## API Programing

- Added Auto-Completion Favorites. When calling a method that contains arguments that are chosen from a number of available options, the options have traditionally been shown in a drop-down list as you type. Now, as you choose an option the first time, it will move to the top of the list and be shown with an \* to indicate that it is a “favorite”. This means that when you come to another argument of the same type, your “favorites” will be easier to find at the top of the list and you won't have to search the entire list to find them. “Favorites” persist for an entire session, but not across sessions.
- Added Symbol Highlighting. After you have created a script, as you click on a parameter, method or other symbol in the script, the User Interface will highlight all occurrences of that symbol throughout the script making it easy to see where that symbol is referenced. In addition, a second scroll bar will appear in the upper right portion of the window. This scroll bar can be used to jump to the next or previous occurrence of the symbol in the script.
- Updated to show the *Watch* window as the default tab when debugging.

## Geometry

- Added ability to remove loops from multiple disconnected faces (i.e., disjoint solids).
- Updated *Duplicate Materials* option in *Other Entities to Include* section to be *Duplicate Materials/Layups* for commands on the *Geometry, Copy...* menu. Same as the *Duplicate Properties* option, but for laminate and solid laminate properties, all referenced layups and all materials used by referenced layups will be duplicated. Any property assigned as a meshing attribute to the original geometric entity will also be copied, then the copy of the property referencing copies of the material(s) and layup will be assigned to the copied geometric entity.

- Updated *Duplicate Materials* option in *Other Entities to Include* section to be *Duplicate Materials/Layups* for commands on the *Geometry, Rotate...* menu. Same as the *Duplicate Properties* option, but for laminate and solid laminate properties, all referenced layups and all materials used by referenced layups will be duplicated. Any property assigned as a meshing attribute to the original geometric entity will also be copied, then the copy of the property referencing copies of the material(s) and layup will be assigned to the copied geometric entity.
- Updated *Duplicate Materials* option in *Other Entities to Include* section to be *Duplicate Materials/Layups* for commands on the *Geometry, Reflect...* menu. Same as the *Duplicate Properties* option, but for laminate and solid laminate properties, all referenced layups and all materials used by referenced layups will be duplicated. Any property assigned as a meshing attribute to the original geometric entity will also be copied, then the copy of the property referencing copies of the material(s) and layup will be assigned to the copied geometric entity.

## Meshing

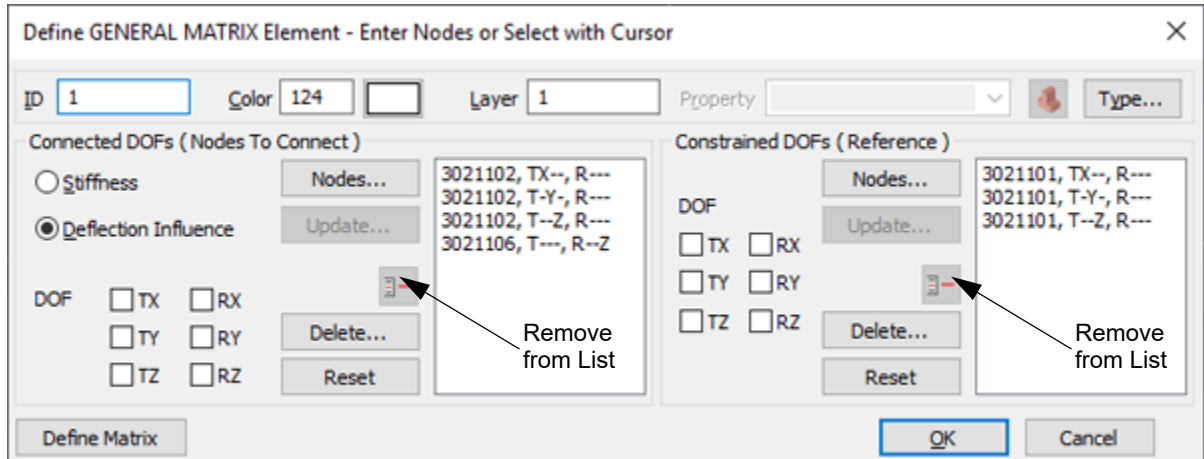
- Added *Set Default Size* button to *Mesh, Mesh Control, Default Size* command. When clicked, the value for *Element Size* is calculated based on all geometry currently in the model and the *Set Element Size on Next Use* option is automatically disabled.
- Updated *Duplicate Materials* option in *Property/Material Options* section to be *Duplicate Materials/Layups* for commands on the *Mesh, Copy...* menu. For laminate and solid laminate properties, all referenced layups and all materials used by referenced layups will be duplicated. When *Geometry* option in *Other Entities to Include* section is enabled, all duplicated laminate and solid laminate properties which have been assigned as a meshing attribute will reference a copy of the referenced layup and copies of all materials used by a referenced layup.
- Updated *Duplicate Materials* option in *Property/Material Options* section to be *Duplicate Materials/Layups* for commands on the *Mesh, Rotate...* menu. For laminate and solid laminate properties, all referenced layups and all materials used by referenced layups will be duplicated. When *Geometry* option in *Other Entities to Include* section is enabled, all duplicated laminate and solid laminate properties which have been assigned as a meshing attribute will reference a copy of the referenced layup and copies of all materials used by a referenced layup.
- Updated *Duplicate Materials* option in *Property/Material Options* section to be *Duplicate Materials/Layups* for commands on the *Mesh, Reflect...* menu. For laminate and solid laminate properties, all referenced layups and all materials used by referenced layups will be duplicated. When *Geometry* option in *Other Entities to Include* section is enabled, all duplicated laminate and solid laminate properties which have been assigned as a meshing attribute will reference a copy of the referenced layup and copies of all materials used by a referenced layup.
- Updated various commands which split elements on the *Mesh, Editing...* menu to more fully and consistently handle the other entities associated and/or related to the elements being edited
- Improved the *Mesh, Geometry, HexMesh Solids* command to provide a warning if you attempt to remesh solids that are already meshed. The warning gives the option to delete the existing mesh, skip the meshed solids, or create a duplicate mesh. This workflow now matches the one for tet meshing.
- Improved performance and accuracy when using length-based mesh sizing on the Parasolid curves (i.e., curves on sheet solids, solids, and general bodies). In a large model with many nonlinear curves, this resulted in a 2X increase in performance and more consistent constant length spacing throughout the model. Also, improved accuracy when attempting to size curves by *Element Size* instead of *Number of Elements* on curves which were seemingly the same length. Due to numerical accuracy, even a very small difference in the length of two curves could result in the curves having a different number of elements assigned. This improvement should improve sizing on curves, surfaces, and solids, whether they are sized by menu commands or *Meshing Toolbox*.

## Elements

- Added *Nastran General Matrix* element type, which is used to support GENEL elements for Nastran solvers. All element information is stored on the elements, thus a corresponding property is not required.

This element type allows you to specify a “general element”, GENEL, for Simcenter Nastran or MSC Nastran, which allows stiffness or deflection influence coefficients to be input for pairs of nodes and their associated DOFs:





Common controls for both *Connected DOFs (Nodes to Connect)* and *Constrained DOFs (Reference)* sections:

The *Nodes* button is used in conjunction with *DOF* check boxes to choose which nodal degrees-of-freedom should be included in the list for that section. *Update* can be used to update the *DOF* for any number of entries currently highlighted in the corresponding list. Use the *Remove from List* icon button to remove any number of selected entries in the multi-select list. The *Delete* button can be used to select any number of nodes to remove from a list using the standard entity selection dialog box. At any time, the *Reset* button to clear the entire list. When you highlight a node in the list, it will highlight in the graphics window, based on the current settings of the *Window, Show Entities* command. See Section 6.3.2.3, "Window, Show Entities..." for more details.

The *Connected DOFs (Node to Connect)* section contains a radio button which determines if the coefficients defined with the *Define Matrix* button are *Stiffness* ("K" rows) or *Deflection Influence* ("Z" rows) values. The nodes and corresponding degrees-of-freedom in this list are written as "UI" and "CI" values to the GENEL entry.

The list in *Constrained DOFs (Reference)* section contains nodes and corresponding degree-of-freedom which will be written as "UDj" and "CDj" values to the GENEL entry.

The *Define Matrix* button displays the *Define Matrix Components for General Matrix Element* dialog box:

3021102, 1	3021102, 2	3021102, 3	3021106, 6
3.767294E-7			
-1.409488E-7	-1.368224E-5		
6.762251E-7	-6.484613E-6	-3.196705E-7	
5.758509E-6	7.472653E-7	1.288941E-6	1.748521E-6

This grid represents the lower triangular portion of the stiffness or deflection influence coefficient matrix. The number of columns is determined by the number of entries in the *Connected DOF (Nodes to Connect)* list and the column headers are labeled using the Node ID and degree-of-freedom of each entry. The cells represent entries in the "N x N" matrix, which are written as "KZij" values to the GENEL entry. When there are a large number of values in the dialog box, it can be resized to make the values fully visible.

## Properties

- Added *Write Zeros at End B (Off=Blank)* option to the Beam Property, which is only for Nastran solvers. This option exists because as a value of 0.0 for an End B value is interpreted differently by Nastran then a “blank” field. When the option is enabled, values of 0.0 specified for End B in the FEMAP property are written as 0.0 to the Nastran input file. When disabled, values of 0.0 are left blank in the Nastran input file.

## Layups

- Added *Send to Excel* button, which opens Microsoft Excel and transfers the current contents of the *Layup Editor* directly to Excel.

## Loads and Constraints

- Added the ability to create *Follower Force* and *Follower Moment* loads on nodes, points, curves, and surfaces.

When creating follower forces or follower moments on nodes, the only available option in *Direction* is *Magnitude Only*. Enter a value for *Magnitude*, then select *Node 1 (G1)* and *Node 2 (G2)* that specify the direction to create FORCE1 or MOMENT1 entries for Nastran. To create FORCE2 or MOMENT2 entries instead, enable the *Normal to Plane* option, then specify *Node 1 (G1)* through *Node 4 (G4)*. The direction of the force or moment is parallel to the cross product of vectors from *Node 1 (G1)* to *Node 2 (G2)* and *Node 3 (G3)* to *Node 4 (G4)*.

When creating follower forces or follower moments on geometry, the process is similar, only 2 or 4 points are selected instead of nodes.

## Connections (Regions, Properties, and Connectors)

- Added Traction Removal Convergence to *Multistep Structural (401)* tab, which is accessed by clicking the *More Options* button.

**Traction Removal Convergence** - Creates the DCTOL field on the BCTPARM entry. Stabilization damping traction removal convergence value used when Stabilization Damping Option (CTDMAP) is set to anything other than “0..No Stabilization Damping” (Default = 0.01)

- Updated *\*Initial Penetration* option on *Multistep Structural (401)* tab by adding “1..Ignore Gaps” option.

**\*Initial Penetration** - Creates the INIPENE field on the BCTPARM entry. Use when the goal is for a pair of contact regions to be touching without interference, but due to the faceted nature of finite elements around curved geometry, some of the element edges or faces may have a slight gap or penetration. (Default = 0)

“0..Calculated” - Contact is evaluated exactly as the geometry is modeled. No corrections will occur for gaps or penetrations.

“1..Ignore Gaps” - Only Gaps will be ignored.

“2..Calculated/Zero Penetrations” - Penetrations will be reset to a new initial condition in which there is no interference.

“3..Zero Gap/Penetration” - Gaps and penetrations are both reset to a new initial condition in which there is no interference.

- Added *Include Shell Thickness Offset* option to *Multistep Kinematic (402)* tab.

**Include Shell Thickness Offset** - When enabled, writes nothing to BCTPAR2 entry, which means shell thickness and Z-offset are both included in the contact problem. This option is only for shell elements referencing the PSHELL property, but not laminate properties. When disabled, writes value of 1 to BCTPAR2 entry, meaning Shell thickness and Z-offset are not included, and contact is evaluated at the midplane of shell elements.

## Functions

- Added “46..True Stress vs. Strain” and “47..True Stress vs. Plastic Strain” for Simcenter Nastran SOL 401/402, as well as “48..Shear Damage vs. Thermodynamic Force (TABLEM5)” for MATDMG entry.

## Aeroelasticity

- Added ability to import Aeroelastic forces, pressures, and coefficients results from .f06 file from Simcenter Nastran and MSC Nastran, then display those results as contour/criteria plot on the aero mesh (aero boxes).

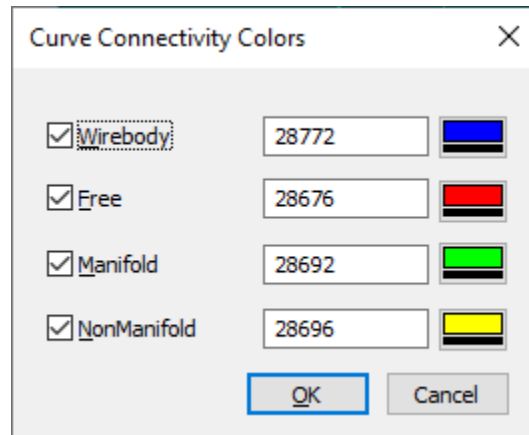
## Listing

- Added *List, Output, Results to Excel* command. See Output and Post-Processing for more information.
- Updated *List, Geometry, Curve* command to include Arc/Circle center coordinates when *Advanced* is chosen.

## Views

- Added functionality throughout *View Options* dialog box which automatically switches the *Color Mode* to “1..View Color” after the user clicks the “Color Block” icon to access the *Color Palette*, then clicks *OK*.
- Added “Element - Nastran General Matrix” option in “Labels, Entities and Color” Category of *View, Options* command, which controls if symbols are drawn to indicate if nodes are specified as *Connected DOFs (Nodes to Connect)* or *Constrained DOF (Reference)*. In addition, *Label Model* allows *Degrees of Freedom* to be labeled at nodes used by Nastran General Matrix elements.
- Added “Curve Connectivity” option in “Tools and View Style” Category of *View, Options* command, which displays the type of connectivity for each curve used by a surface (sheet solid), solid, or general body using a color specified by the user.

Clicking the *Connectivity Colors...* button will open the *Curve Connectivity Colors* dialog box:



The three types of curve connectivity are:

**Free** - all “edge curves” which are only used by a single surface. “Free curves” in a solid usually indicate “gaps” or “holes” in the geometry, meaning the solid does not fully enclose a volume and is probably not viable for solid meshing (tet or hex). If multiple surfaces are “stitched” together but do not enclose a volume (sheet solid) or “joined” using the *Geometry, Surface, NonManifold Add* command (general bodies), then “free curves” may also indicate “gaps” or “holes” between surfaces. Of course, “free curves” in this type of geometry may be internal holes/loops or the outside edge of the stitched/joined “part”, which are normal.

**Manifold** - all “edge curves” which are being shared by more than one surface. Typically, this is the case when multiple surfaces are “stitched” together to enclose a volume (solid) or are “stitched” but do not enclose a volume (sheet solid). In addition, includes “edge curves” of surfaces which are “joined” using the *Geometry, Surface, NonManifold Add* command (general bodies), but does not include any “edge curves” of one surface which are shared with “internal curves” of another surface (i.e., “T-junctions”).

**NonManifold** - all “nonmanifold edges” in the geometry. Only geometry that has been joined using the *Geometry, Surface, NonManifold Add* command (general bodies) or imported geometry which contain general bodies will have these type of edges. Typical “nonmanifold edges” are found where surfaces come together at “T-junctions” or a surface has been “NonManifold added” to a Solid

In addition, this option also displays curves considered **Wirebody** entities by Parasolid with a unique color

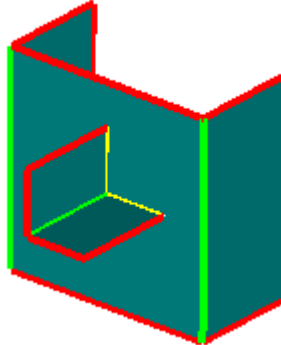
**Note:** Wirebody entities cannot be created in FEMAP, they can only be imported, therefore it is not common for a curve to be considered a wirebody entity.

The display of each individual type of connectivity can be enabled/disabled using the check box to the left of name and the “color block” button to the right can be used to select a color using the standard *Color Palette* dialog box.

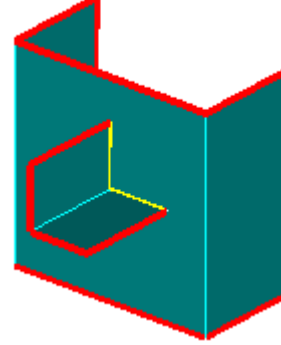
Display when option disabled



Display with option enabled



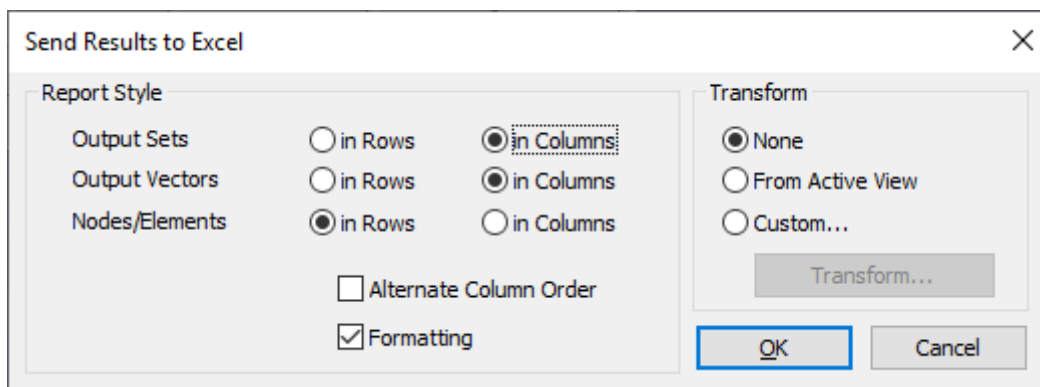
Display of Free and NonManifold



## Output and Post-Processing

- Added *List, Output, Results to Excel* command.

...similar to *List, Output, Results to Data Table*, provides the ability to quickly send chosen output vector values for specific nodes or elements (from any number of selected output sets) directly to Microsoft Excel using a specified “Report Style”.



Other than the destination of the output values, two major differences between this command and *List, Output, Results to Data Table* are:

1. Both Nodal and Elemental data can be sent to Excel in a single command with Nodal output values being sent to one worksheet, while Elemental values are set to a different worksheet in the same workbook.
2. The *Formatting* option. When *Formatting* is enabled, which is the default, the following formatting items are specified in Excel to improve the user experience:
  - the color of each row below the column header alternates between white and light blue
  - column headers are shown with **BOLD** type
  - each column automatically has a filter applied
  - a frozen cell is specified on each worksheet to keep column headers and at least first cell of each row visible

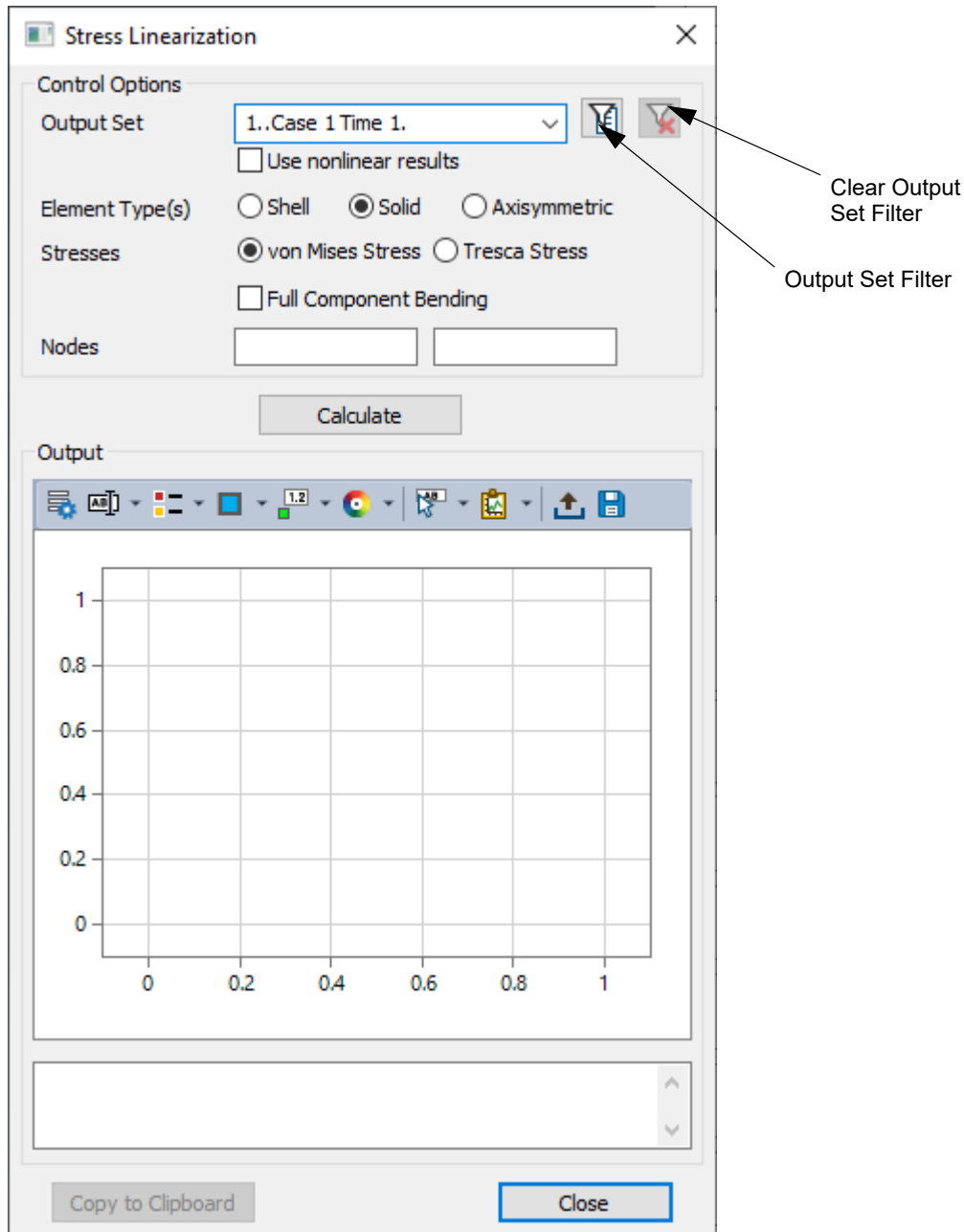
	A	B	C	D
1	ID	1..Simcenter NASTRAN	1..Simcenter NASTRAN	1..Simcenter NASTRAN
2	245	1242.061646	8.039933205	36.05553818
3	246	1322.185547	13.68164825	59.51531219
4	247	1447.109253	61.8647995	123.9129333
5	248	-96.36251831	1583.993164	-170.6546021
6	249	44.67827606	1072.014771	-63.40066147
7	250	1154.833008	29.01952362	72.66993713
8	251	47.76041794	1034.34436	-54.84164047
9	252	1153.972778	49.00695038	99.79524994
10	253	1089.015503	70.3221283	87.01507568
11	254	-206.9236755	918.8821411	51.42808914
12	255	981.6740112	-27.67558861	-3.919765472
13	256	-184.1565399	1007.183594	21.35284233
14	257	1024.072144	44.06726074	35.07287979
15	258	997.1751709	13.67201138	16.66805267
16	259	992.2723999	-59.73863602	-6.34272337
17	260	-165.7936096	992.3218384	10.93315887
18	261	1007.018921	36.80402756	10.10761547
19	262	992.3222046	-1.570205688	4.220077515
20	263	989.9287109	-71.65684509	-2.943143606
21	264	-156.154892	997.6192017	3.148905993
22	265	44.67887115	1072.014526	63.40047073

When *Formatting* is disabled, none of the additional formatting is done, which allows the command to be used simply as a tool to quickly transfer output values from FEMAP to Excel as “raw data”.

- Added View, Advanced Post, Stress Linearization command.

... allows you to perform Stress Linearization. The FEMAP Stress Linearization Tool is based on the ASME Boiler and Pressure Vessel Code, 2007 SECTION VIII, DIVISION 2, ANNEX 5.1 LINEARIZATION OF STRESS RESULTS FOR STRESS CLASSIFICATION (moving forward, referred to simply as “ASME Code”).

This command opens the *Stress Linearization* dialog box:



### Understanding Stress Linearization

The ASME Code, and this tool, are based on the user choosing an appropriate “Stress Classification Line” (SCL) by choosing two nodes.

**Note:** It is imperative that users be familiar with the requirements and recommendations of the ASME Code, as the proper choice of the SCL is key to applying this standard. If the standard is not applied according to the ASME Code, then the output of this tool will likely not be useful.

Once chosen, stress tensors are interpolated from the selected output data at points along the SCL. Stresses are then transformed/aligned with the SCL. The Membrane Stress Tensor is calculated by integrating the individual stress tensor components along the SCL. The Bending stresses are integrated along the SCL as well, and, per ASME code, the transformed axial stress aligned with the SCL. The two shear stress tensor values perpendicular to this transformed axial stress are not used in the bending tensor calculation. There is option to include all stress tensor components in the bending tensor calculation provided in FEMAP, *Full Component Bending*, but again, is not recommended by the ASME Code.

**Note:** Stress Linearization relies on transforming FEA tensor stresses to be aligned with the SCL. For this calculation to be correct, the reference coordinate system for the stress data must be known. The base reference system can be specified in the FEMAP user interface by either using *File, Preferences*, choosing the *Geometry/Model* tab, then clicking the *Output Orientation* button or using the *Transform* button found in various dialog boxes, then clicking the *Output Orientation* button. In all cases, the *Current Output Orientation* dialog will be displayed to set the base orientation.

**It is the responsibility of the user to verify the original reference system of the stress data.**

### Using the Stress Linearization tool

The *Stress Linearization* dialog box is designed to allow the user to specify all options, choose the nodes for the SCL, and examine a plot of results in a single place.

#### Control Options

**Output Set** - allows selection of a single output set to use as the initial data for the calculations.

**Note:** If the data needed to perform the calculations is not available in an output set, a message of “Stress Linearization Failed” will be sent to the message box below the chart in the *Output* section.

**Output Set Filter icon button** - opens the *Filter Output Set* dialog box, which can be used to reduce the number of output sets which appear in the *Output Set* drop-down list. To filter, enter text into the *Title Includes* field to reduce the list to only include output sets using the entered text in their Titles. Output sets can also be filtered by selecting a *Study*, *Analysis Program*, and/or *Analysis Type*. Click the “Output Set Filter” again and enter more text to further reduce the list.

**Clear Filter icon button** - click to return all output sets to the *Output Set* drop-down list.

**Use nonlinear results** - when enabled, uses nonlinear results in the selected output set instead of linear results.

**Note:** If using an output set from a nonlinear analysis that does not have both linear and nonlinear output vectors, this option is not available, as only nonlinear output exists in that output set.

**Element Type(s)** - choose which type(s) of elements, *Shell*, *Solid*, or *Axisymmetric*, to retrieve results for the calculations. The nodes selected for the SCL must be from the same type of element and those elements must reference the same material.

**Stresses** - choose either *von Mises Stress* or *Tresca Stress* to use for the calculations.

**Note:** Stress is calculated from the interpolated stress tensor as either *von Mises* or *Tresca*. Tresca is a failure criterion based on Max Shear. The Tresca “Stress” calculated/displayed is the maximum principal stress minus the minimum principal stress, which is two times maximum shear.

**Full Component Bending** - when enabled, the two shear stress tensor values perpendicular to the transformed axial stress are used in the bending tensor calculation.

**Note:** The use of the *Full Component Bending* option is NOT recommended by the ASME Code.

**Nodes** - these fields are used to specify the two nodes which define the SCL

**Note:** Selected nodes must be on same type of element and the elements must reference the same material.

#### Calculate button

Once all options are specified and nodes for SCL are selected, click this button to populate the *Output* section.

#### Output

This section contains an embedded chart which displays the results of the stress linearization calculations for specified SCL. It also includes a message window containing useful information such as which nodes define the SCL, the type of stresses were used in the calculations, and the maximum stress values.

The embedded chart uses the same underlying technology as the *Charting* pane, but only offers a subset of the features and options. For instance, because all of the data being plotted on embedded chart is calculated on-the-fly, it uses only a single chart and there is no need to create any individual data series.

Available features to modify the default options used by the embedded chart are:

- icon menus to specify options for *Chart Title*, *Chart Colors*, and *Show Tooltips*, along with *Copy to Clipboard* icon menu to send chart data and/or the chart image to the clipboard or send both directly to Microsoft Excel
- individual context-sensitive menus for the overall *Chart*, the *X Axis* and *Y Axes*, the *Legend*, the *Title* (when visible), and individual curves which are currently plotted
- ability to double-click in the overall chart or on the *X Axis*, *Y Axis*, the *Legend*, the *Title* (when visible), and individual curves which are currently plotted to open familiar dialog boxes from the *Charting* pane

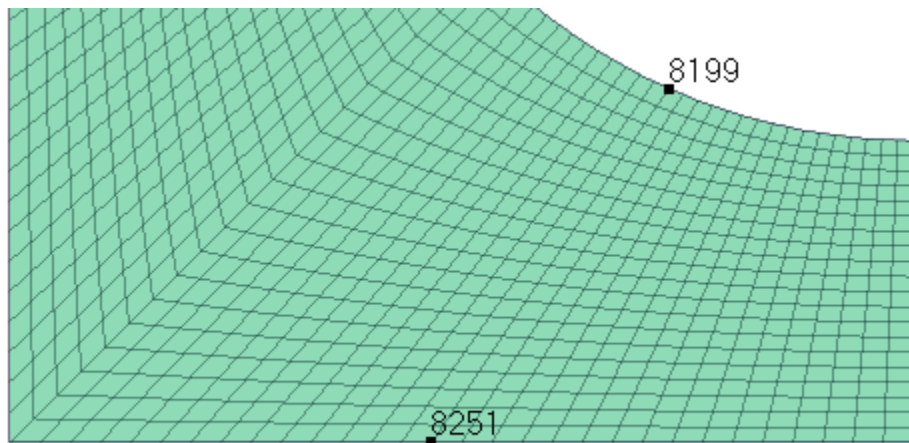
For more information about the functionality of the *Charting* pane, see Section 7.2.4, "Tools, Charting"

### Copy to Clipboard

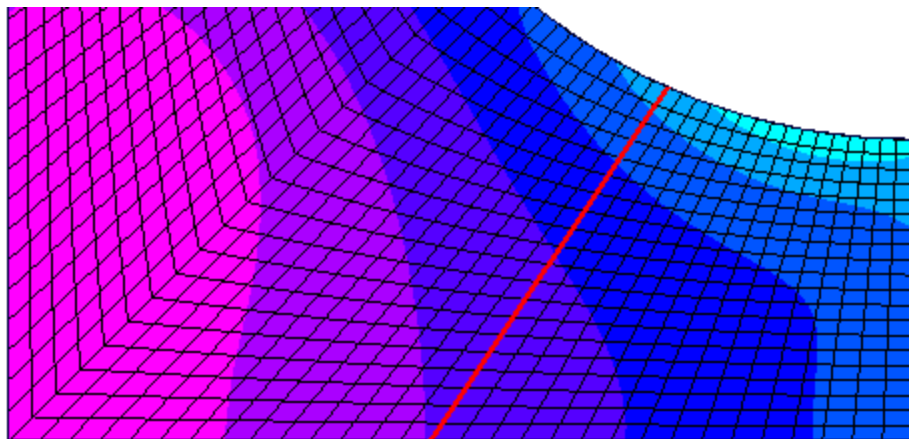
Clicking this button will send a copy of the chart and some additional information, such as which nodes define the SCL, the type of stresses used in the calculations, maximum stress values, and how elemental stresses were transformed for each element topology.

### Example

Nodes of Axisymmetric elements used for Stress Concentration Line (SCL):



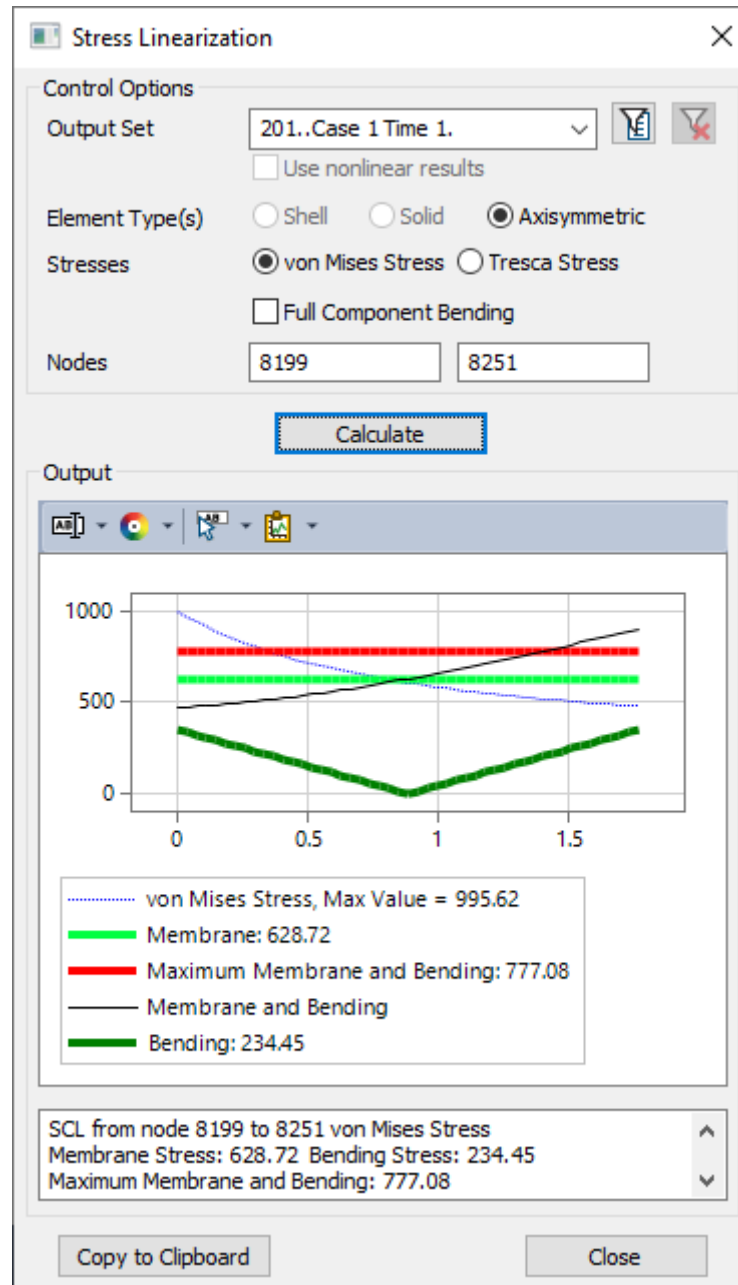
Model is axisymmetric elements with mapped meshed (SCL Shown in Red):



Output Set: Case 1 Time 1.  
Elemental Contour: Axisym Von Mises Stress



Results of Stress Linearization calculations:



Example of listing sent to clipboard by *Copy to Clipboard* button (not icon menu):

**Stress Linearization**

SCL from node 8199 to 8251 von Mises Stress

Membrane Stress: 628.72; Bending Stress: 234.45

Maximum Membrane and Bending: 777.08

Axisymmetric Stresses Transformed from NASTRAN Standard, Radial, Azimuth, Axial to the SCL

- Added ability to display Aeroelastic forces, pressures, and coefficients results as a contour/criteria plot on the aero mesh (aero boxes).

- Added support for results generated by Simcenter Nastran for progressive ply failure (Damage Status, Damage Values, Damage Energy, and Crack Density).
- Improved performance of *Copy*, *Linear Combination*, *RSS Combination*, and *Envelope* operations in the *Model*, *Output*, *Process* command. For *Linear Combination*, *RSS Combination*, and *Envelope*, this should reduce the processing time by at least 25%. In addition, improves performance when accessing output sets which were created with the *As Needed/Temporary* option via the *Result Set Processing Table* (or *Data Surface*).

## Geometry Interfaces

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

FEMAP Interface	Latest Supported Version
Parasolid	32.1
NX	version 1899 (2020)
SolidWorks	2020
Pro/Engineer	Pro/Engineer 16 - Creo 6
CATIA V5	V5 R8 - V5-6 R2019 SP4
JT	10.6

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 v2020.2 changes

### Simcenter Nastran Interface (Previously NX Nastran)

#### SOL 401 and SOL 402

- Added read/write support for MATDMG entry to specify material properties for progressive ply failure using two material types found in *Other Types*. To create a MATDMG with PPFMOD set to “UD”, use “512..Sim. Nastran UD Ply Failure (MATDMG Sol 401,402)” which references a 3D Orthotropic material. To create a MATDMG with PPFMOD set to “EUD”, use “513..Sim. Nastran EUD Ply Failure (MATDMG Sol 401,402)” which references a 3D Orthotropic material.
- Added support to automatically write PFRESULTS to request progressive ply failure results when a material which writes a MATDMG entry to the input file exists in the model.

#### SOL 401 Only

- Added support for *LGSTRN* option in *NASTRAN Bulk Data Options* dialog box, which writes PARAM,LGSTRN,1 to input file. When enabled, *LGDISP* option becomes unavailable.
- Added support for *Rot. Load Inertia Scaling (RFVAR)* option in *Solution and Convergence Options* dialog box.

**Rot. Load Inertia Scaling (RFVAR)** - Specifies how the software interpolates the time variation for the rotational loads RFORCE or RFORCE1. Writes RFVAR field with selected option (Default = “0..Scale Angular Velocity and Acceleration”).

*0..Scale Angular Velocity and Acceleration* - interpolates time variation using  $f(t)$ . As a result, if a linear time variation is defined, the angular velocity will vary linearly. Since “omega” is squared in the force computation, the force will not vary linearly. Writes OMEGA.

*1..Scale Force* - interpolates time variation using  $f(t)^2$ . The software then takes the square root of the interpolated value before applying it to the angular velocity. As a result, if a linear time variation is defined, the force will vary linearly, but “omega” will not. Writes FORCE

#### **SOL 402 Only**

- Added support to write TRUE to STRMEAS field on MATS1 entry by using a function of type “46..True Stress vs. Strain” or “47..True Stress vs. Plastic Strain” for *Function Dependence on Nonlinear* tab for a material.
- Added support for reading results generated by the SHELLTHK Case Control entry.
- When *LGSTRAN* option is enabled, *LGDISP* option now becomes unavailable.

A number of bugs were corrected.

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

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

- Added read/write support for GENEL entry as Nastran General Matrix element type.
- Added support to write values specified as 0.0 for End B of beam properties as either 0.0 or “blank” fields to PBEAM entries, based on each property’s setting for *Write Zeros at End B (Off=Blank)*.
- Added support to write AEROF and APRES case control commands for Static Aeroelastic analysis (SOL 144), by default, which produce aeroelastic force, pressure, and coefficient results in the .f06 file only. In addition, these results can now be imported from the .f06 and shown as contour/criteria plots on aero mesh (aero boxes).

A number of bugs were corrected.

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

### **ANSYS Interface**

- Added ability to import results from compressed ANSYS results files (\*.RST files). Only files compressed with “Version 1” compression, which is the current default in ANSYS, are supported.
- Added *Skip CSys*, *Skip Nodes*, *Skip Materials*, *Skip Properties*, and *Skip Elements* to *Model Control* section of *ANSYS Command and Model Options* dialog box. When enabled, the entity type selected to “skip” will not be written to the ANSYS input file. For example, if *Skip Nodes* and *Skip Properties* are enabled, then no nodes or properties will be exported.
- Added *Skip Groups* to *Model Control* section of *ANSYS Command and Model Options* dialog box. When enabled, groups will not be written to the ANSYS input file (ESEL, NSEL, and CM entries), but entities contained in those groups are still exported unless one of the other Skip options is enabled.
- Added *Skip Misc* to *Model Control* section of *ANSYS Command and Model Options* dialog box. When enabled, comments starting with a single “!”, view settings (/VIEW, /ANGLE, /ZOOM, etc), preprocessor commands (/NOPR, /FCOMP, /PREP7, /GOPR, etc), solution setting (/SOLU, ANTYPE, EQSLV, etc), SOLVE, and FINISH are not written to the ANSYS input file.
- Updated name of *Skip Beam/Bar Cross Sections* in *Model Control* section of *ANSYS Command and Model Options* dialog box to *Skip Shapes*. When enabled, all Beam and Bar properties are written to the ANSYS input file as SECTYPE, #, BEAM, ASEC, along with the corresponding computed property values from the *Define Property - BEAM Element Type* dialog box as SECDATA, regardless of how the beams were defined.
- Added *File Compression Level* option to *Result Control* section of *ANSYS Command and Model Options* dialog box, which sets the level of compression for the ANSYS Results file (\*.RST). Default in FEMAP is “1..No Compression”, but other option is “0..Sparse”, which creates a compressed results file FEMAP can read.
- Added *Manual Control* sections containing a *Skip Standard* option, along with *Start Text* and *End Text* buttons to all appropriate ANSYS dialog boxes in the Analysis Set Manager.
- Improved performance between 2x-3x when importing non-compressed ANSYS results files (\*.RST files).

A number of bugs were corrected.

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

### **ABAQUS Interface**

- Added support to read loads and boundary conditions which reference a SET and create a Load Definition or Constraint Definition in FEMAP.
- Added support to write all supported load/constraint types that are contained in a Load Definition or Constraint Definition as a SET entry in the ABAQUS input file, if requested by enabling the *Write All Groups as Sets* option in the *ABAQUS Model Options* dialog box. When expanded, some Load Definitions which were defined with a single value actually create variable values on entities, such as geometry-based loads which are expanded to nodes. Because the values are not constant, they cannot be written as a single SET entry, thus the loads are exported on a per entity basis, as they were in previous versions of FEMAP.

A number of bugs were corrected.

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

### **LS-DYNA Interface**

- Added *Skip CSys*, *Skip Nodes*, *Skip Materials*, *Skip Properties*, and *Skip Elements* to *Model Control* section of *LS-Dyna Model Options* dialog box. When enabled, the entity type selected to “skip” will not be written to the LS-Dyna input file. For example, if *Skip Nodes* and *Skip Properties* are enabled, then no nodes or properties will be exported.
- Added *Skip Groups* to *Model Control* section of *LS-Dyna Model Options* dialog box. When enabled, groups will not be written as \*SET\_ entries to the LS-Dyna input file, but entities contained in those groups are still exported unless one of the other Skip options is enabled.
- Added *Skip Functions* to *Model Control* section of *LS-Dyna Model Options* dialog box. When enabled, functions (\*DEFINE\_CURVE entries) are not written to the LS-Dyna input file. In addition, any loads or material properties which were functionally-dependent in the model no longer reference functions
- Added *Skip Misc* to *Model Control* section of *LS-Dyna Model Options* dialog box. When enabled, comments starting with “\$”, \*KEYWORD, \*TITLE with corresponding “title”, \*CONTROL TERMINATION with corresponding value, \*DATABASE\_BINARY\_D3PLOT with corresponding value, \*END, and other entries are not written to the LS-Dyna input file.
- Added *LS-Dyna Analysis Monitor* to monitor solver progress, review solver files, and optionally import results.

A number of bugs were corrected.

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

### **Tools**

- Updated the *Tools, Parameters* command to allow specification of the Nodal Output Coordinate System (*Output CSys* drop-down). While this could be done using the *Model, Node* command and pressing *Parameters* button or in other meshing commands by pressing *Node Parameters* icon button, this is more convenient.
- Updated *Tools, Mass Properties, Mesh Properties* command to be able to automatically send the mass properties listing to the clipboard using the *Send to Clipboard* option. In addition, Check Mass Properties dialog box has been reorganized into separate *Reporting Options* and *Mesh Options* sections.
- Updated the *Tools, Mass Properties, Solid Properties* command to be able to calculate and output the volume, surface area, center of gravity, and moment of inertia for the selected sheet solid(s), solid(s), and/or general bodies, either as a single assembly or for each individual body. Additional options exist to create a representative mass element and potentially attach it to existing nodes.

Once the solid(s) have been selected, the *Check Mass Properties* dialog box is displayed:

### Reporting Options

The *Reporting Options* section contains a drop-down to specify which *Coordinate System* will be used for the mass properties listing and *Send To Clipboard* option. When *Send to Clipboard* is enabled, a similar listing to what is sent to the *Messages* window and other destinations specified by the *List, Destination* command (see Section 7.5.9, "List, Destination..."), will also be sent to the clipboard, making it easy to paste into another application.

### Mass Calculation

This section contains options used to calculate mass properties from the selected solid(s), with two main options:

***Compute Overall/Assembly Properties*** - when selected, overall mass properties for all selected sheet solids, solids, and/or general bodies will be computed as if they are a single assembly/body.

***Compute Properties for Individual Solids*** - when selected, individual mass properties will be calculated for each selected sheet solid, solid, and/or general body.

The remainder of the section is used to specify how the density and/or thickness information is provided for the calculations. If the selected bodies have meshing attributes specified, the *Use Density/Thickness form Meshing Attributes* option will be enabled by default.

When enabled, all density and thickness information used for the mass properties calculations is retrieved from the properties and materials referenced by the mesh attributes.

If disabled, or no meshing attributes has been assigned to the selected bodies, then values for *Density for Solids with No Meshing Attributes* and/or *Thickness for Surfaces with No Meshing Attributes* must be provided by the user. If desired, the *Material* icon button can be used to retrieve a density from an existing material or the *Property* icon button can be used to retrieve a thickness from an existing property.

Example of listing when using *Compute Overall/ Assembly Properties* option:

```

Messages
Check Solid Mass Properties
3 Solid(s) Selected...
Solid Mass Properties in CSys 0..Global Rectangular
Density = 0.000733145   Solid 1..Untitled
Density = 0.000733145   Solid 2..Untitled
Density = 0.000733145   Solid 3..Untitled

Volume      Surface Area      Mass      X      Center of Gravity      Y      Z
3.          18.          0.00219944      1.          0.          0.5

Inertias
About CSys      Ixx      Iyy      Izz      Ixy      Iyz      Izx
0.000916431    0.00458216    0.0040323    0.          0.          0.00109972
About CG        0.000366573    0.00183286    0.00183286    0.          0.          0.

```

### Mesh Options

This section contains options to create different types of entities using the calculated values. When using *Compute Overall/Assembly Properties*, a single node, mass element, and element(s) connecting the mass element to existing nodes can be created for all selected bodies. When using *Compute Properties for Individual Solids*, a single node and mass element can be created for each selected body, but connected to existing nodes.

**Create Node at Center of Gravity** - when enabled, a single node is created at the calculated center of gravity for the entire assembly or each individual body.

**Create Representative Mass Element** - when enabled, which is default when *Create Node at Center of Gravity* is enabled, a mass element which references a mass property containing calculated values for the assembly or each individual body is also created at the calculated center of gravity.

**Link to Mesh** - when enabled, the node created at the center of gravity is attached to any number of existing nodes by a single rigid element or a number of individual elements. The nodes are selected using the standard entity selection box after clicking OK to the Check Mass Properties dialog box.

There are two options for creating the element(s) that link the node at the center of gravity to existing nodes:

**Multi-Element Connection** - when selected, individual elements are created between the node created at the center of gravity and each selected node using the functionality of the *Mesh, Connect, Closest Link* command. For more information, see Section 5.2.3.1, "Mesh, Connect, Closest Link...".

**Single Rigid Element Connection** - when selected, which is the default, a single rigid element is created with the node created at the center of gravity as the *Independent* node and all selected nodes as *Dependent* nodes using the specified *DOF* (degrees-of-freedom).

## Model Merge

- Added ability to access the functionality of the *File, Merge* command via the FEMAP API by using the MergeTool Object. See OLE/COM API for more information.
- Improved handling when using the *File, Merge* command to overwrite Load Sets, Constraint Sets, Connection Regions, Connections, Functions, Tables, Fields, Analysis Sets, Views, Result Sets, Discrete Value Sets, and Matrix Inputs.

## OLE/COM API

### New and modified API Objects and Attributes

- Added MergeTool (feMergeTool) object to the API. Also, added UsePattern, CreateGroupForMerged, AlwaysCreateParentCSys, CondenseMergedGroups, LimitToMergedEntities, KeepInOriginalSets, and DuplicateFirstInPattern to the feMergeTool object.

- Added ndID1, ndID2, ouSetID, nElemType, nStressmode, bNonlinearMode, bFullStressTensor, dMembraneStress, dBendingStress, dMaxMemBend, and dMaxStress to the Stress Linearization (StressLinear) Object.
- Added NasMsnlCntRFVAR attribute for Simcenter Nastran SOL 401 to the Analysis Case Object.
- Added NasMsnlCntRFVAR attribute for Simcenter Nastran SOL 401 to the Analysis Manager Object.
- Added AnsModelGroupID, AnsModelSkipCsys, AnsModelSkipNode, AnsModelSkipMatl, AnsModelSkipProp, AnsModelSkipElem, AnsModelSkipGroup, and AnsModelSkipMisc attributes for ANSYS to the Analysis Manager Object.
- Added DynModelGroupID, DynModelSkipCsys, DynModelSkipNode, DynModelSkipMatl, DynModelSkipProp, DynModelSkipElem, DynModelSkipGroup, and DynModelSkipMisc attributes for LS-Dyna to the Analysis Manager Object.
- Added addl\_ptID and vaddl\_ptID attributes to the LoadGeom Object.
- Added RenderLogDepthOffset attribute to the View Object.
- Added DO\_BeamDiagram\_Option, DO\_BeamDiagram\_RoundToZero, DO\_BeamDiagram\_RoundToZeroV-value, DO\_BeamDiagram\_BeforeDecimalSeparator, DO\_BeamDiagram\_MinimumExponentSize, DO\_BeamDiagram\_ExponentDigits, DO\_BeamDiagram\_SigFigs, DO\_BeamDiagram\_LeadingZeros, DO\_BeamDiagram\_TrailingZeros, DO\_BeamDiagram\_DecimalPlaces, DO\_BeamDiagram\_MaxDigits, DO\_BeamDiagram\_BaseExponent, DO\_BeamDiagram\_OriginalDigits, and DO\_BeamDiagram\_Exponent attributes to the View Object. These attributes are used by the “Beam Diagram” option in *View Options* to control the display of digits in the graphics window for Beam Diagrams.
- Added CurveConnectWirebody, CurveConnectFree, CurveConnectManifold, CurveConnectNonManifold, CurveConnectWirebodyColor, CurveConnectFreeColor, CurveConnectManifoldColor, and CurveConnectNonManifoldColor attributes to the View Object. These attributes are used by the “Curve Connectivity” option in *View Options* to control the visibility and color for the display of curve connectivity.
- Updated RenderPushForward, RenderPushUnit, RenderPushUndeformed, and RenderPushLabel on the View Object. They now have specific values ranges assigned.

### New and Updated API Methods

- Added Clear, SelectModel, SelectModelByName, SelectCurrentModel, GetEntityTypes, GetEntityStatus, GetEntityOptions, SetAllStatus, SetStatus, SetEntityStatus, SelectEntity, SelectFromGroup, ShowDialog, AddRelated, Merge, AlongVector, PointToPoint, BetweenCSys, BetweenVectors, BetweenPlanes, RotateAroundVector, RotatePointToPoint, Reflect, VectorPattern, CSysPattern, and CreatePattern methods to the MergeTool Object.
- Added AddSetFromModel, AddNodesOnFreeEdges, AddNodesOnFreeFaces, and IsIdenticalSet methods to the Set Object.
- Added CalcStressLinearization method to the Stress Linearization (StressLinear) Object.
- Added AnsWriteGroup, AnsSkipCsys, AnsSkipNode, AnsSkipMatl, AnsSkipProp, AnsSkipElem, AnsSkipGroup, AnsSkipMisc, and AnsRSTcomp methods for ANSYS to the Analysis Manager Object.
- Added DynWriteGroup, DynSkipCsys, DynSkipNode, DynSkipMatl, DynSkipProp, DynSkipElem, DynSkipGroup, and DynSkipMisc methods for LS-Dyna to the Analysis Manager Object.
- Added NextOnNode method to the BCEqn Object.
- Added NextOnEntity method to the BCGeom Object.
- Added ManifoldType method to the Curve Object.
- Added GetCentroidArray, GetSingleFaceInfoArray, GetMultiFaceInfoArray, and ElementalCSys methods to the Element Object.
- Added NextOnEntity method to the LoadGeom Object.
- Added NextOnEntity method to the LoadMesh Object.
- Added ComputeStdShape2 and ComputeGeneralShape2 methods to the Property Object.

- Updated EraseSet method to the Draw/Erase Object.

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

- feAppGetModelByName
- feAppRunningApplicationInfo
- feAppGetRunningApplication
- feEdgesOfFreeFaces

**New and updated Global Variables**

- Added Pref\_RenderCtrlGGroupEvaluate, Pref\_RenderDisableAbort, Pref\_OpenMP, and Pref\_OpenMPThreads to set preferences on the *Graphics* tab of *File, Preferences*. Also, updated Pref\_RenderRotate to be able to control the *Center* option in the *Include In Dynamic Rotation* section.
- Added Pref\_GlobalLanguage to set *Language* option and Pref\_GlobalUIScaling to set *UI Scaling* option in the *Global Options* section of *User Interface* tab of *File, Preferences*
- Added Pref\_Abaqus\_PreserveCompressedLoads to set *Preserve Load/BC SET(s)* on *Interfaces* tab of *File, Preferences*
- Added Pref\_ComputeAverageMidResults to set *Compute Averaged Mid Stress/Strain* preference on *Results* tab of *File, Preferences*
- Added ModelFromPreviousVersion, ModelMigratedFromVersion, and ModelMigratedFromDB as new Global Parameters which can determine from which previous version the model was migrated from and other information.
- Updated Pref\_RenderRotate and vPref\_RenderRotate to be able to control *Center* option in *Included In Dynamic Rotation* section of *Graphics* tab.
- Updated Pref\_EntityColor and vPref\_EntityColor to be able to control *Mesh Point* and *Monitor Point* color in *Entity Colors* section of *Color* tab.

## Preferences

### Graphics

- Added *Ctrl-G Group Evaluate* to *Graphics Options* section.

By default, the *Group, Operations, Evaluate* command (see Section 6.4.3.2, "Group, Operations Menu") is not performed on any groups when *Window, Regenerate* (see Section 6.3.2.2, "Window, Regenerate...") is used (*Ctrl-G* shortcut key). When this option is enabled, *Group, Operations, Evaluate* is performed on all groups currently visible in the active view when *Window, Regenerate* is used. If *View, All Views* (Section 6.1.2, "View, All Views...") is enabled, then all groups currently visible in all views of the active model will be evaluated. If *Show Full Model* is specified for *Groups* in the *View, Visibility* dialog box, the *Model Info* tree, or the *Grp:* menu in the *Status Bar*, no groups are evaluated.

- Added *Disable Abort* to *Graphics Options* section.

Certain operations can cause the graphics window to be completely redrawn without explicitly using *Window, Redraw*. In some cases, the user can abort (interrupt) the redraw by clicking in the screen or selecting another command. For this to happen, FEMAP checks for user input during redraw, which can cause the redraw to take additional time to finish compared to not doing any checking. When this option is enabled, no checking is done, thus aborting a redraw is not possible and potentially improving performance.

- Added *Center to Include In Dynamic Rotation* section.

When enabled, the *Center* option displays a symbol in the graphics window at the "center of rotation" currently being used for dynamic rotation. If the "center of rotation" has been set by *View, Rotate, Rotate About Rotation Center*, then the "center of rotation" is obvious. Alternatively, if using the "view center" and the *Dynamic Rotate Around Cursor Location* preference found on the *User Interface* tab is enabled, then it may be helpful to have a visual representation at the "center of rotation".

- Added *OpenMP* and *Options...* button to *Advanced/Debug Options* section.



**This is initial implementation of OpenMP for multi-threading, thus it is considered as Beta Functionality which is not officially supported. If any issues are encountered, please disable this option, then feel free to contact GTAC to describe the issue.**

Enable this option to use OpenMP functionality (i.e. multi-threading) which has been implemented for a limited number of commands and operations. Once enabled, click the *Options...* button to set *Number of Threads*, which should not exceed the number of cores available on the machine running FEMAP.

### User Interface

- Added *Global Options* section containing two new options, *Language* and *UI Scaling*.

#### Language

Language displayed by user interface. When set to “0..Default”, uses the language which was selected during the initial installation of that version of FEMAP. When set to any other option, that language is used by FEMAP. Available option are “1..English”, “2..German”, “3..Japanese”, “4..Chinese, Traditional”, or “5..Chinese, Simplified”.

**Note:** If the selected language is not installed and/or properly specified using a number of Windows settings on a computer, then most, if not all text will be unreadable throughout the user interface.

#### UI Scaling

When set to “0..Enabled”, which is the default, certain Windows display settings are used to determine how the user interface should be scaled to improve user experience. Scaling the user interface can be especially helpful when using a high resolution display. If scaling is not required for any reason, set this option to “1..Disabled”.

### Interfaces

- Added *Preserve Load/BC SET(s)* option to *ABAQUS Options* section.

Sometimes in an ABAQUS input file, \*SET entries with unique names are used to specify “groups” of loads or boundary conditions. When this option is enabled, an attempt is made during import of the ABAQUS input file to collect loads or boundary conditions defined by a \*SET entry into an appropriate Load Definition or Constraint Definition.

### Results

- Added *Compute Average Mid Stress/Strain* option to *General Solver Options* section.

When enabled, stresses and strains at middle locations for plate elements are calculated as a simple average of the top and bottom values in FEMAP.

**Note:** Many solvers allow the user to recover “top” and “bottom” results which are NOT located at +/- 0.5\*Element Thickness, thus are NOT actually at the top and bottom of the element. No attempt is made to interpolate these type of results to the actual middle of the element.

### Color

- Added *Monitor Point to Entity Colors* section to control default color of monitor points.

