

# FEMAP v2019.1 New Features and Corrections

## *Updates and Enhancements*

### **Views**

- Added ability to renumber Views using the Modify, Renumber, View command. Also, Views can now be renumbered using the Modify, Renumber, All command.
- Updated “Label Parameters” option in “Labels, Entities and Color” Category of View, Options command by replacing the Label Font box and the Performance Graphics Font button with two drop-downs in the Font section, which are used to specify the font and size to use for both entities supported by performance graphics and those that are not currently supported (i.e., all text in the view is shown using a single “unified font”).
- Updated “Performance Graphics” option in “Tool and View Style” Category of View, Options command by removing the Performance Graphics Font button, as all text in the view is shown using a single “unified font”.
- Updated “Contour/Criteria Style” option in “PostProcessing” Category of View, Options command by adding the “2..Contour with Zero” option to Elements with no results. Using “2..Contour with Zero” can aid in locating contact results which only exist on particular faces, but are obscured by other elements.
- Added “Region” option in “PostProcessing” Category of View, Options command, which controls if results on nodes or elements should be visible “through” any type of displayed region when a contour/criteria plot is also being displayed. In the case of Connection Regions, this can be quite useful for the results to be displayed “on top of” regions, especially when the analysis included any kind of contact. When enabled, which is the default, the Contour/Criteria option allows results to be visible “through” the displayed regions, while Label Mode controls if the output values will be visible through the displayed regions as well (“1..Allow Output Values”, default), or not (“0..No Labels”).

### **Analysis Manager**

- Added Delete icon buttons to the Select Optimization Variable(s), Select Topology Region(s), Select Manufacturing Constraint(s), and Select Optimization Limit(s) dialog boxes in the Analysis Set Manager.

### **Connection Properties, Regions, and Connectors**

- Updated the Bolt Region dialog box by adding the Axes Determined by Solver option to the Solid Bolt Options section. This option is only available when Bolt Type is set to Solid. When enabled, the Bolt Axis CSys and Dir controls become unavailable and can be helpful when creating certain types of bolts for Simcenter Nastran.
- Updated the titles of the tabs in the Define Connection Property dialog box used to specify connection property values for Simcenter Nastran (formally NX Nastran) to Linear, Multistep Structural(401), Multistep Kinematic (402), Adv Nonlin (601), and Explicit (701).

- Updated the look and feel of the ANSYS tab by isolating the Friction Coefficient (MU), placing all the Real Constants in the upper portion of the dialog box, and all of the Contact KEYOPTs in the lower portion.
- Added ABAQUS Thermal... button to ABAQUS tab, which displays the Define Gap Options dialog box. See Properties section for a description of the options in the Define Gap Options dialog box.
- Added options to the Multistep Structural (401) tab, which are accessed by clicking the More Options... button. The new options are \*Adaptively Modify Penalty Factor, \*Adjustment Tolerance, \*Const Offset Distance, \*End Time for Initial Penetrations, and \*Sliding Contact Formulation.

## Geometry

- Updated Modify, Project, Point command by adding two options to the Direction section, Radial Around Axis, and Radial Around Point, which project points outwards from either a defined vector or specified location.
- Updated functionality of operations which update geometry, specifically ones which split existing geometry into smaller pieces, to attempt to automatically update geometry-based loads, constraints, and/or regions.

## GUI - Toolbars and Icons

### View and View - Simple Toolbars

- Added Feature Lines toggle to View Style menu of the View and View - Simple toolbars. Used enable and disable the display of feature lines, which are otherwise controlled by the Feature Line option in the Tools and View Style category of the View Options dialog box.

### Select Toolbar

- Added Locate in Model Info toggle to Selector Modes menu of the Select toolbar. Only available when Select Single Item is the active setting in the Selector Modes icon menu. When Locate in Model Info mode is activated, the Model Info tree pane is currently visible, and the active entity is set to Solid, CSys, Property, Material, or Layup, the currently selected entity will also be highlighted in the Model Info tree.
- Added Select Visible Only toggle to Selector Actions menu of the Select toolbar. Allows you to select only from entities which are currently “visible” in the active view

## GUI - Dockable Panes

### Model Info Tree

- Added Copy and Renumber commands on context-sensitive menu for Views, which are used to copy or renumber, respectively, any number of views currently highlighted in the Model Info tree.
- Updated Compare command on the context-sensitive menu for Results by asking for confirmation before comparing a large number of output sets to one another, as each output set is compared to every other selected output set, which in some cases will require a large amount of time.

## Meshing Toolbox - General

- Updated functionality of operations which update geometry, specifically ones which split existing geometry into smaller pieces, to attempt to automatically update geometry-based loads, constraints, and/or regions.

## Charting Pane

- Added Show When Selected icon buttons to the Data tab for all data series types which allow selection of nodes and/or elements, as well as groups.
- Updated the location of certain fields and control on the Data tab for all data series types other than “0..Vector vs. Entity” and “5..Function”.
- Added ability to renumber Charts and Chart Data Series from the Modify, Renumber menu. Also, Charts and Chart Data Series can now be renumbered using the Modify, Renumber, All command.

## Entity Editor

- Added Material CSys item for Solid, Solid Laminates, and Solid Cohesive Elements which have a material coordinate system override (MATCID) assigned, which can be updated from the Entity Editor.

## Data Table

- Added Previous On icon buttons to the Results to Add to Data Table dialog box, which is accessed via Add Output Columns icon.
- Added Material CSys item for Solid, Solid Laminates, and Solid Cohesive Elements which have a material coordinate system override (MATCID) assigned.

## **Interfaces - FEMAP Neutral**

- Updated Neutral Read and Write for v2019.1 changes

## **Interfaces - Nastran**

- Added ability to import Aero Mesh deformations from Flutter Analysis from .f06 file (only noticeable difference to the user is that new output vectors are available).
- Added STATSUB Options dialog box for buckling subcases.
- Added more generalized read support for THRU fields for SPC1, ASET1, BSET1, etc entries, as long as the form “a” THRU “b” is used and all three fields reside on a single line, as it cannot continue across lines.
- Added Contact section to NASTRAN Output Requests dialog box and added options for Contact and Glue.
- Added “4..Frequency/Harmonic Response” to the Analysis Type drop-down in the NASTRAN Optimization Options dialog box used to setup SOL 200 Design Optimization. When defining optimization limits for “4..Frequency/Harmonic Response”, the Frequency Response - Nodal Displacement, Frequency Response SPC Forces, and/or Frequency Response Element Force, Stress, Strain options must be used.

- Added Buckling option to the NASTRAN Modal Analysis dialog box which can be accessed in an analysis set which has Analysis Type set to “10..Nonlinear Static”. When Buckling is enabled, the Inverse Power/Strum method will be selected automatically and some additional options are available. Writes PARAM,BUCKLE,2, which requests buckling in a SOL 106 cold start run, is written along with an EIGB entry.

## **Interfaces - Simcenter Nastran (formally NX Nastran)**

- Updated NX Nastran to Simcenter Nastran throughout the FEMAP application.
- Added the ability to specify a material coordinate system override (MATCID entry) for solid, solid laminate, and solid cohesive elements, which is done via the Modify, Update Elements, Solid Material CSys command.
- Added support for glued contact in SOL 200.
- Added “4..Frequency/Harmonic Response” to the Analysis Type drop-down in the NASTRAN Optimization Options dialog box used to setup SOL 200 Topology Optimization. When defining optimization limits for “4..Frequency/Harmonic Response”, the Frequency Response - Nodal Displacement, Frequency Response SPC Forces, and/or Frequency Response Element Force, Stress, Strain options must be used.

### SOL 401 and 402 Only

- Added ability to choose “3..Transient” for Analysis Type for the Master Case and Subcases
- Added Create Initial Conditions From Output icon button to Boundary Conditions dialog box

### SOL 401 Only

- Added Arc-Length Control Options dialog box to specify parameters written to the NLARCL entry.
- Added Mass and Damping dialog box to specify mass and damping values for dynamic analysis.
- Added Time Integration section to Solution and Convergence Options dialog box.

### SOL 402 Only

- Added ability to choose “7..Buckling” for Analysis Type for Subcases. When used, adds a Modal item to tree for the subcase, which opens the NASTRAN Model Analysis dialog box.
- Added Viscous Material Option section of the Multi-Step Control Options dialog box.
- Added options to the Time Step section of the Multi-Step Control Options dialog box. Also, updated many of the names of the other options in the section to match what is now expected by Simcenter Nastran.
- Added options to the Analysis Control section of the Multi-Step Control Options dialog box.
- Added option to the Other Options section of the Multi-Step Control Options dialog box.

## **Interfaces - MSC Nastran and Autodesk Nastran**

- Added support to read and write pyramid elements

## Interfaces - ANSYS

- Added ability to set up a random response analysis by selecting “6..Random Response” from the Analysis Type drop-down in the Analysis Set dialog box. Options for random response are specified in the ANSYS Modal Analysis Options dialog box and the new ANSYS Random Vibration Analysis Options dialog box.
- Added Mass Matrix (LUMPM) and Damping Matrix sections to the ANSYS Modal Analysis Options and ANSYS Harmonic Analysis Options dialog boxes.
- Added support to read and write bolt regions and bolt preloads for ANSYS. Bolt Regions create PSMESH and PRETS179 entries, while Bolt Preloads create SLOAD entries.
- Added support to read and write non-normal pressure loads on 2D/3D elements edges/faces. Distributed non-normal pressures are mapped to ANSYS input files as SFE command together with SURF153, SURF154 or SURF156 elements.
- Added Skip Beam/Bar Cross Sections option to ANSYS Command and Model Control dialog box. When enabled, writes all beam and bar properties 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 support to read and write KEYOPT(8) for laminate and solid laminate elements. This value is stored as a formulation specify this value in FEMAP, use the Modify, Update Elements, Formulation command.

## Interfaces - ABAQUS

- The Microsoft MPI (Message Passing Interface) is required to attach to Abaqus ODB files. This interface is no longer included during the FEMAP installation. Beginning with FEMAP 2019.1, Siemens PLM Software can no longer redistribute the Microsoft MPI. Anyone who requires Abaqus ODB files in their workflow will need to take extra steps to ensure the necessary MPI is installed. The download and installation instructions are available from Microsoft using the following link: <https://www.microsoft.com/en-us/download/details.aspx?id=57467>
- Added support to read and write \*GAP CONDUCTANCE, \*GAP CONDUCTANCE, PRESSURE, and/or \*GAP RADIATION entries. To create or edit these values in FEMAP, use the ABAQUS Thermal... button in the Define Property - GAP Element Type dialog box or the on the ABAQUS tab of the Define Connection Property to open the Define Gap Options dialog box. For more information, see the Properties section.

## Interfaces - Geometry

- Added support for Parasolid 31.1, NX 2019, and SolidWorks 2019.

## Node - Project

- Updated Modify, Project, Node command by adding two options to the Direction section, Radial Around Axis, and Radial Around Point, which project nodes outwards from either a defined vector or specified location.

## **Element - Formulations**

- Added options to the Ansys Keyopt section on Ansys tab of the Element Formulations dialog box to specify where results should be recovered for laminate and solid laminate elements. Sets the value of KEYOPT(8). For linear laminate elements, use the fourth drop-down in Ansys Keyopt section. For parabolic laminate elements set to “1..SHELL91” or “2..SHELL99”, use first drop-down or use the second drop-down when set to “0,,SHELL281”. For linear or parabolic solid laminate elements, use the third drop-down.

## **Element - Updates**

- Added Modify, Update Elements, Solid Material CSys command, which allows a material coordinate system override to be specified for solid, solid laminate, and/or solid cohesive elements
- Added Modify, Update Elements, Connect Linear/Parabolic Elements command, which updates the nodes on linear elements to match the nodes on parabolic elements which are connected to the linear elements or vice versa.

## **Properties**

- Added the ABAQUS Thermal... button to the Define Property - GAP Element Type dialog box

## **Layups**

- Added ability to specify Ply Failure Theory ply-by-ply to support PCOMPG1 for SOL 401 and 402.
- Added ability to specify Ply Failure Theory when creating or editing a Global Ply.

## **Loads and Boundary Conditions**

- Added the ability to create Bearing Force and Torque loads on curves.

## **Optimization**

- Added Frequency Response - Nodal Displacement, Frequency Response SPC Forces, and Frequency Response Element Force, Stress, Strain options to the Category drop-down in the Optimization Response dialog box used to define optimization limits. These options must be used to properly set optimization limits when Analysis Type is set to “4..Frequency/Harmonic Response” in the NASTRAN Optimization Options dialog box in the analysis set.
- Added Delete icon buttons to the Select Optimization Variable(s), Select Topology Region(s), Select Manufacturing Constraint(s), and Select Optimization Limit(s) dialog boxes in the Analysis Set Manager.

## **Meshing**

- Updated the Mesh, Extrude, Element and Mesh, Extrude, Element Face commands by adding the Along Normal to Surfaces option to the Method section of the Generation Options dialog box. This options is similar to using Along Vector to Surfaces when planar elements are selected, but uses each element’s normal direction as the extrusion vector instead of requiring a vector to be specified.

- Updated all commands on the Mesh, Extrude and Mesh, Revolve menus by adding the Extrude in Both Directions option. This option works a little different for each method, but essentially it creates elements along or around the specified vector(s) as well as in the opposite direction(s).
- Updated the Mesh, Geometry, Solids command by adding the ability to automatically create pyramid elements in transition areas when Meshing Approach is set to Tet/Pyramid Mesh. In addition, added the Pyramid Mesh Options section to the dialog box to control certain aspects of the automatically generated pyramids.
- Updated all commands on the Mesh, Sweep menu by adding a Methods section to the Generation Options dialog box. The Along Curve method allows these commands on the Mesh, Sweep menu to be used as they have existed in FEMAP for many releases, while the Along Element Edges method offers new functionality. The Along Element Edges method also introduces a new dialog box to facilitate the selection of element edges on both planar and solid elements.

## **Listing**

- Consolidated the functionality of the List, Output, Standard and List, Output, Use Format command into a single command, List, Output, Formatted, which lists output data using a format from a library or a custom format saved with the model.
- Updated the name of List, Output, Format command to List, Output, Custom Format Definition and moved it into a different section of the List, Output menu.

## **Aeroelasticity**

- Updated the name of List, Output, Format command to List, Output, Custom Format Definition and moved it into a different section of the List, Output menu.

## **Output and Post-Processing**

- Added ability to import the deformations generated by a aeroelastic flutter analysis for the aero mesh on aero panels and aero bodies, then display those deformations using the various Deformed Style options available in the View Select dialog box or specified via Style drop-down in the Deform tool of the PostProcessing Toolbox.
- Added ability to display results “on top of” regions, which can be quite useful attempting to view results from an analysis included any kind of contact.

## **Groups and Layers**

- Added the Group, Element, using Orientation Node command, which will add any element which uses a selected node as an orientation node to the active group.

## **Tools**

- Added the Tools, Structural Analysis Toolkit command, which simply opens the Structural Analysis Toolkit. This command is only available when the Structural Analysis Toolkit is installed and properly licensed.

- Updated the Tools, Parameters command by adding Mesh Point to specify Color, Next ID, and Inc for mesh points. In addition, added Layup and Function to specify Next ID and Inc for layups and functions, respectively.
- Updated the Tools, Text command by replacing the single Font drop-down, which offered only 77 combinations for font and, with two drop-downs, one to select the Font and the other to specify the size of the font.

## Libraries

- Updated libraries in a number ways, including a new dialog box with various tabs and controls which offer expanded functionality, storing of “personal” libraries in the user directory instead of the FEMAP installation directory, and a new preference to specify a single location for libraries “shared” with other users.

## User Interface - General

- Added Select Visible icon button to the standard entity selection dialog box. In addition, replaced Reset button with a Reset icon button (Red Circle with White “X”) and added an icon image to Select All button.
- Added Select Visible Only option to the Pick^ menu in the standard entity selection dialog box. When Select Visible Only is disabled, which is the default, any entity in the model which fit the criteria entered for a By... method or are related via an Add... option will be selected. When Select Visible Only is enabled, only entities which are currently visible in the active view and fit the criteria or are related will be selected. The methods on the Pick^ menu which only consider “visible” entities when Select Visible Only is enabled are By Size, By Color, By Model Data Value, By Output, Add Connected Tangent Curves, Add Connected Fillets, Add Tangent Surfaces, Add Connected Elements, and Add All Connected Elements.
- Added using Orientation Node to the Method^ menu of the standard entity selection dialog box when selected elements, which selects any element which uses a selected node as an orientation node.

## Preferences

### Database

- Added Windows Explorer Data section. This section controls if a thumbnail image and/or general information about the model is stored with a FEMAP model file (\*.modfem). Save Thumbnail, enabled by default, will store an image (active view of model when model is saved) with the model, which is displayed with/as the icon for \*.modfem file in Windows (File) Explorer. Save Size and Notes Info, enabled by default, stores information including: FEMAP Version number; Number of Solids, Nodes, Elements, and/or Output Sets; and the first 10 lines of the text specified for the model via the File, Notes command. This information will be displayed in a tooltip when the cursor is placed over a \*.modfem file for a short period of time.

### Geometry/Model

- Added Solid Boolean Tolerance option to the Geometry Preferences section. Controls the tolerance value the Parasolid geometry kernel uses for Boolean operations. By default, the value is 0.0, which



automatically determines a tolerance based on the Solid Geometry Scale Factor. **Changing this setting may cause geometry problems. Do not change this setting unless instructed to do so by FEMAP Support**

#### Library/Startup

- Moved Material Type Definition library to new Other Libraries section.
- Updated the name of the Libraries section to Startup Personal Libraries (FileNames Also Set Standard Shared/System Libraries).
- Added Startup Shared Libraries section.

## API

#### New and updated API Objects and Attributes

- Added NasMsnlArcLenOn, NasMsnlArcLenConstr, NasMsnlArcLenMinAlr, NasMsnlArcLenMaxAlr, NasMsnlArcLenMaxR, NasMsnlArcLenScale, NasMsnlArcLenDIter, NasMsnlArcLenMxInc, NasMsnlArcLenInitLdfac, NasMsnlArcLenMxLdfac, NasMsnlArcLenSkipFac, NasMsnlCntTINTMTH, NasMsnlCntBETA, NasMsnlCntGAMA, NasMsnlCntALFA, NasMsnlCntTETA, NasStatsubOn, NasStatsubBuckle, NasStatsubPreload, NasMsnlkCnt2MATSYM, NasMsnlkCnt2INERTIA, NasMsnlkCnt2CRINFAC, NasMsnlkCnt2ITEREF, NasMsnlkCnt2TSDYN, NasMsnlkCnt2RJPZ, NasMsnlkCnt2RJPN, NasMsnlkCnt2TSVSC, NasMsnlkCnt2VSCOTE, and NasMsnlkCnt2VSCOSN attributes to the Analysis Case Object. Also, added NasMsnlkCnt2DTINIT, NasMsnlkCnt2DTMIN, NasMsnlkCnt2DTMAX, NasMsnlkCnt2EQMFMX, NasMsnlkCnt2EQMFMIN, and NasMsnlkCnt2TINTMTH, which were added to support some options which were renamed in Simcenter Nastran.
- Added NasBulkPARAMBuckle, NasMsnlCntTINTMTH, NasMsnlCntBETA, NasMsnlCntGAMA, NasMsnlCntALFA, NasMsnlCntTETA, NasMsnlkCnt2MATSYM, NasMsnlkCnt2INERTIA, NasMsnlkCnt2CRINFAC, NasMsnlkCnt2ITEREF, NasMsnlkCnt2TSDYN, NasMsnlkCnt2RJPZ, NasMsnlkCnt2RJPN, NasMsnlkCnt2TSVSC, NasMsnlkCnt2VSCOTE, and NasMsnlkCnt2VSCOSN attributes to the Analysis Manager Object. Also, added NasMsnlkCnt2DTINIT, NasMsnlkCnt2DTMIN, NasMsnlkCnt2DTMAX, NasMsnlkCnt2EQMFMX, NasMsnlkCnt2EQMFMIN, and NasMsnlkCnt2TINTMTH, which were added to support some options which were renamed in Simcenter Nastran.
- Added UseSolverAxes attribute to the Connection Region Object.
- Added items to the flag and pval attributes of the Connection Property Object (for ABAQUS Thermal options of ABAQUS tab and new options on Multistep Structural tab).
- Added CopyToActiveLayer attribute to the CopyTool Object.
- Added location attribute to the GFXArrow Object.
- Added failuretheory attribute to the Global Ply Object.
- Added failuretheory and vfailuretheory attributes to the Layup Object.
- Added items to the flag and pval attributes of the Property Object (for ABAQUS Thermal options for Gaps).
- Added ShowInModelInfo and SelectVisibleOnly attributes to the Selector Object.

## New and Updated API Methods

- Updated GetLibrary, PutLibrary, and DeleteLibrary methods on the common entity object to allow selection of the type of library, “Personal”, “Shared”, or “Femap Standard Libraries”.
- Updated GetLibraryOfType method on the Material and Property objects.
- Added SPOPT, PSDUNIT, PFACT, and PSDCOM methods to the Analysis Case Object.
- Added GetUsedFREQS, PutUsedFREQS, SPOPT, PSDUNIT, PFACT, and PSDCOM methods to the Analysis Manager Object.
- Added AddDataColumn and GetColumnInfo2 methods to the Data Table Object.
- Added GetMatlOrientVec as a method to the Element Object
- Added GetFREQType, SetFREQData, GetFREQData, SetFREQ1Data, GetFREQ1Data, SetFREQ2Data, GetFREQ2Data, SetFREQ3Data, GetFREQ3Data, SetFREQ4Data, GetFREQ4Data, SetFREQ5Data, GetFREQ5Data methods to the Frequency Object. Also, updated Get and Put methods.
- Added AddPly2, InsertPly2, SetPly2, GetPly2, SetAllPly2, and GetAllPly2 methods to the Layup Object. Also, updated HasGlobalPly method.
- Added RemoveNotVisible and AddAllAnalysisCases methods to the Set Object.
- Added SetFontData and GetFontData methods to the Text Object. Also, updated the Get method.
- Added CollectorAppearanceSetFontData method to the User Defined Graphics Object.
- Added SetFontData and GetFontData methods to the View Object. Also, updated ElementNoResultMode to accept an additional value.
- Added Spin method to the View Orient Object.

## New and Updated Global Variables

- Added Pref\_ModelThumbnail, Pref\_ModelTooltip, PickVisibleOnly, Pref\_SharedLibPath, Pref\_GeometryBooleanTolerance, Pref\_GraphicsFont, and Pref\_GraphicsFontSize to set various preferences.
- Added Pref\_SuppressScreenEntities, which can disable selection of all “Active Screen Entities” (i.e. View Legend, View Axis, Post Titles, and Contour Legend) in the graphics window.
- Added PickVisibleOnly option which enables the Select Visible Only option on the Pick^ menu of the standard entity selection dialog boxes and on Selector Actions icon menu of the Select toolbar.
- Updated Pref\_MsgWndFontSize and Pref\_MsgWndFontSize to set various preferences.

## Removed Global Variables

- Removed AllOn, AllOff, TurnOn, TurnOff, AddFreq, AddFreq1, AddFreq2, AddFreq3, AddFreq4, AddFreq5, AddFreqByFuncID, AddFreq5ByFuncID, UpdFreq, UpdFreq1, UpdFreq2, UpdFreq3, UpdFreq4, UpdFreq5, UpdFreqByFuncID, UpdFreq5ByFuncID from the Frequency Object

The following functions have been added or updated:

- feGetCurrentLibraryName
- feSetCurrentLibraryName
- feSolidInsideMulti
- feMeshTetSolid3
- feModifyEdit

- feNodesBetweenNodes
- feSolidElementsInARow

## ***Corrections***

### **General**

- Corrected an issue that could cause a loss of some data and/or options when renumbering Analysis Sets for certain Analysis Types.
- Corrected issue when renumbering Functions when they reference other Functions.

### **Analysis Manager**

- Corrected issue in where the first item in the Response Frequencies list on the Solution Frequencies tab of the NASTRAN Dynamic Analysis dialog box could be displaying the wrong frequency value.
- Corrected issue when copying optimization analysis and/or analysis cases containing optimization entities which caused separate references to optimization objects (variables, manufacturing constraints, limits) to not be created correctly.

### **Connection Properties, Regions, and Connectors**

- Corrected issue which caused the corners of the Contact Box on Connection Regions to be transformed, even if no Contact Box was previously defined for the region. When the Contact Box is not defined, the corners are normally both set to the global origin. Previously, if the region was moved, the corners were both moved. They still defined an empty box and did not normally cause any issues, but it was preferred to leave them at the origin (PR# 9451923)

### **Geometry**

- Corrected an issue that caused repetitive error messages to be displayed in the various Move commands when attempted to move part of a solid, since only complete solids can be moved. Now, only a single warning is displayed.
- Corrected issues which could cause geometric “washers”, created by the Geometry, Curve - From Surface, Offset Curve/Washer command or the Geometry Editing tool in the Meshing Toolbox, to fail at larger values specified for Solid Geometry Scale Factor (i.e. 1000.0).
- Corrected issue in Geometry, Solid, Slice which could occur when Method is set to “with Curves” and the “Along Curve Normal” and/or the “Vector Direction Only” option(s) are enabled, where the solid would be partially sliced.
- Corrected an issue when using the Geometry, Solid, Add command where the “Ok to create a Non-Manifold Solid” message was presented in error.

### **Graphics**

- Corrected issue where entities in connection regions would obey their individual layer ID and not the contact region layer ID (PR# 9302160).

- Corrected issue where Aero Spline on CAERO2 panel not spaced correctly. Spline was spaced evenly instead of using the CAERO2 chord points.
- Corrected issue where Aero Spline labels were draw at origin for some flat panels. Also moved spline label so it does not overwrite panel label.
- Corrected issue where Feature Lines and curves were not depth sorted correctly. This occurred whether Performance Graphics/Best Possible is enabled or not.
- Corrected issue where deleting surfaces from a solid did not update the “view volume”.
- Corrected issue where free face and free edge did not handle missing mid-side nodes correctly, which caused extra free faces and free edges, which were incorrect.
- Corrected issue which cause the View Legend to move when clicking on the View Axes.
- Corrected issue which allowed graphical picking of a Property instead of an Aero Property when creating an Aero Panel/Body.
- Corrected issue in multi-set animation when the selected contour data does not exist for one frame, which caused the contours and contour legend to disappear from all following frames in frame creation.
- Corrected issue where Minimum and Maximum values were not drawn in second cycle of multi-set animation, therefore these values did not appear in the animation replay.
- Corrected issue where Nonlinear Force loads were not drawn in groups. They are now drawn in groups based on the ID of the node specified in the Apply Force To section.
- Corrected issue where warning/error messages are issued when post processing certain types of output on three-nodes elements, when transformed. For some results, Nastran does not have corner results for three-noded elements, which was causing the incorrect warning/error messages.
- Corrected issue with highlighting front pick of surfaces for general beam section evaluation. No highlighting would show until the model had been rotated.

## **Performance Graphics**

- Corrected issue with RBE2 and RBE3 element contour and criteria (and criteria labels).
- Corrected issue which caused incorrect stress contours to be displayed on parabolic wedge elements, when the type of element was either parabolic solid or parabolic cohesive. The third mid-side node was not averaged correctly.
- Corrected issue where dynamic rotation would not use the closest location on the model (i.e., when the Dynamic Rotate Around Cursor Location option was enabled in Preferences) when overall transparency is enabled and/or individual entity transparency is specified.
- Corrected issue with material direction was not being drawn correctly for Solid Laminates or Solid Cohesive elements.
- Corrected issue which caused coordinate systems to not be displayed with the correct “lighting” if the View Axis was not being drawn.
- Corrected issue which caused all Distributed Loads on line elements to not be visible when using Intel graphics.
- Corrected issue which caused any Distributed Load on a beam element to be scaled incorrectly when scaling all by maximum value.
- Corrected issue which caused any Distributed Load on a line element to not be drawn at all if the value at End A or End B was specified as zero.

- Corrected issue when drawing Y axis for CBUSH and Curved Tubes.
- Corrected issue where criteria labels were not drawn in performance graphics when limit mode is set to no limits and elements that pass draw entity is off. Labels should be controlled just by the elements that pass label mode.

## GUI - General

- Corrected an issue that prevented selection of Custom Output Formats via a Title List in the standard selection dialog box.
- Corrected issues in the Modify, Update Elements, Property ID command; the Mesh, Edge/Skin Elements commands; the Mesh, Mesh Control, Attributes at Point command; and the Mesh, Mesh Control, Attributes on Solid command. Previously, when the dialog to select a property was initially displayed, it only contained Properties of compatible types, however if a new Property was created, the list would change to show all Properties. It now always shows only compatible Properties. Also, updated the dialog to immediately select and highlight any newly created Property.
- Corrected an issue that occurred when using the Workplane... command on the Quick Access Menu (i.e., the right mouse button context menu) while in the standard Entity Selection dialog. If choosing to define the Workplane using the standard Specify Plane dialog, multiple coordinates would be filled by a single click in the graphics window.
- Corrected an issue that occurred if all model files were closed, then the File, Open command was used, instead of the File, Import command, to open a FEMAP Neutral or Parasolid file. Previously, this would result in the Minimize, Restore, and Maximize buttons at the end of the Menu Bar for this model to be hidden.
- Corrected issue where preview of a plane or vector when selecting a saved plane or vector would not draw on screen as reversed if the "Reversed" check box was selected.
- Corrected issue which would sometimes cause dialog box used to specify Element Formulations from an older version of FEMAP to be displayed instead of the current dialog box.

## GUI - Toolbars and Icons

### Select Toolbar

- Corrected an issue that resulted in the Select Toolbar being turned off if the "By Size" command in the Selector Action Menu and measured an entity while in the command.

## GUI - Dockable Panes

### Model Info Tree

- Corrected several issues with Nonlinear Force Loads in the Model Info tree and the related context sensitive menu commands. Previously, if the loads were not in a Load Definition, the Edit commands displayed an incorrect dialog box. If a Load Definition contains a single Nonlinear Force Load, the "Edit Where Applied" and "Edit Load" commands both now bring up the appropriate Nonlinear Force editing dialog box. If the Load Definition contains multiple Nonlinear Force Loads, both commands display an error message informing the user to split the definition into individual loads prior to editing.

- Corrected an issue that occurred if “View All Layers” was enabled and either the “Show Selected” or “Hide Selected” context menu for the layer check boxes was used. Previously, only the selected check boxes were made available and all others remained grayed and unusable. Now all check boxes are made available.

#### Meshing Toolbox - Feature Removal Tool

- Corrected an issue which could occur when using the Feature Removal Tool to remove Loops on a Surface and the Add Surface Mesh Point option was enabled. Previously, if the Loop represented a hole and another Solid passed thru the hole, the Mesh Point was incorrectly associated with the Solid rather than the original Surface.

#### Meshing Toolbox - Geometry Editing Tool

- Corrected an issue which occurred when Operation was set to Project/Move Point and a point on a general body solid was moved, which caused the geometry and mesh to not be updated properly.

#### Charting Pane

- Corrected issue where the help documentation for charting would not appear when F1 was pressed if the charting pane had focus, rather than the charting control
- Corrected an issue where saving a chart object in the API did not update the chart selection drop-down in the Charting Pane
- Corrected issue where charting tooltips would show the incorrect data type when hovering over a point when Convert in the Output Processing section was set to 0..Average or 1..Max Value.
- Corrected issue in Charting dialog where multiple radio buttons would be selected for Legend Direction if loading a chart from the library where the legend direction was set to Left to Right”

#### Data Table

- Corrected issue that would cause performance problems in the List, Output, Results to Data Table command.

### **Interfaces - FEMAP Neutral File**

- Corrected an issue that prevents additional master case options from being stored in the database when reading neutral files.

### **Interfaces - Nastran**

- Corrected an issue that would cause separate initial conditions, such as displacement and velocity, respectively, in the form of Loads on the same node/element ID to not be written in a TIC bulk data entry (i.e., Displacement for each of the components of the nodes specified in 6 different Loads would result in only one TIC entry for that node).
- Corrected issue which caused the “Bush/Fast Z Force” and “Bush/Fast Z Moment” output titles to not be correct when reading Nastran op2 files.
- Corrected issue when writing Aerodynamics Flutter solutions that caused duplicate PARAM,COUPMASS lines to be written.
- Corrected issue reading Nastran NSM and NSML cards with continuations. (PR# 9294539)

## **Interfaces - Simcenter Nastran (formally NX Nastran)**

- Corrected an issue that caused GAPVAL for the SOL401 BCTPARAM card to not be exported or imported.
- Corrected an issue that prevented initial temperatures from being written with the ID specified in case control, for SOL401, and caused certain TEMP entries to not be written as a result.
- Corrected an issue that would write an error message if bolt preloads defined by strain were applied to Type 3 bolt regions in SOL401.
- Corrected an issue which prevents the BOLTRESULTS case control command from being written if only a bolt sequence had been specified for SOL401.
- Corrected issue which caused Connection Regions which have Type set to Rigid to not be written out properly to use for SOL 402.
- Corrected an issue that caused the wrong default value for TSCEQ on the NLCNTL card to be exported.
- Corrected issue where “checkerboarding” manufacturing constraint on Simcenter Nastran DMNCON card was not being imported correctly.
- Corrected issue where NX Nastran Total and Elastic Strain output was read improperly.
- Corrected issue writing BGSET when both contact and glue Connectors referenced the same Connection Property (PR# 9305558).

## **Interfaces - ABAQUS**

- Corrected issue when reading the \*FRICTION entry from an input file which caused the friction values to not be imported correctly.
- Corrected issue which caused an unneeded Orientation entry to be written when a plate element has plate offsets specified. Will now only export Orientation entry if a Material direction is specified for the element (PR# 9355792)

## **Interfaces - ANSYS**

- Corrected issue with reading SFE command, format error when no space between fields.
- Corrected issue with reading repeat SFE command, old pressure overwritten instead of a new pressure on same element and face.
- Corrected issue with writing of a Nastran beam sections (I, T1, T2, Hat1, L, Box1, Chanel) to ANSYS input file. The orientation, offset, shear center and centroid of these beam sections were not correctly written.
- Corrected issue with reading unused ANSYS element property data. ANSYS element property data, such as beam or plate section data, needs to be interpreted correctly with a suitable element type.
- Corrected issue with interpreting stress and strain components for composite solids, writing an EORIENT command to be consistent with FEMAP post processor.
- Corrected issue when reading ANSYS results for Laminate and Solid Laminate elements.
- Corrected issue with writing SNOPTION command, which caused incorrect characters to be written.

- Corrected issue when reading ANSYS results for Laminate elements which caused the titles of the output vectors to be “Plate Ply#” instead of “Lam Ply#”.

## **Interfaces - LS-DYNA**

- Corrected issue which caused incorrect \*DEFINE\_BOX statements to be written for some connection regions even though they did not have a contact box defined (PR# 9451923)

## **Layups**

- Corrected an issue that causes a control in the Layup Editor to not be updated when a global ply, which a ply in the layup reference, is edited and the user confirms that they want to update the material and/or thickness of the global ply in all layups.

## **Meshing**

- Corrected an issue reflecting plate elements with variable thickness. Previously, the thicknesses were not properly updated to correspond to the change in element normal that occurs during the reflection. Now, thicknesses specified on elements are updated and new properties are created if variable thicknesses were specified on the property.

## **Output and Post-Processing**

- Corrected an issue that occurred when attempting to process multiple “As Needed” (virtual) Output Sets at once. For example, choosing List, Output, Results to Data Table with Multiple “As Needed” Linear Combination Output Sets.
- Corrected issue in freebody validation tool where nodes which are missing SPC, MPC or Applied Load results may not appear in the listing
- Corrected issue in List, Output, Force Balance Interface Load Summary command where an incorrect error message could be displayed when no freebody or no output set(s) were selected
- Corrected issue where, in certain circumstances, freebody nodal vectors could display in the freebody coordinate system or the nodal coordinate system of the previous node when the “User Nodal Output CSys” option is enabled.
- Corrected issue which occurred when Contour Arrows were displayed on line elements using Arrow Display Mode set to Shear (XY) / Axial (Z) and then the List, Output, Contoured Results to Data Table command was used to send the information to the Data Table. Previously, the Axial value was placed into the Shear column and all of the values in the Axial column were set to 0.0.
- Corrected issue which occurred when Contour Arrows were displayed on line elements using Arrow Display Mode set to Automatic or Components and then the List, Output, Contoured Results to Data Table command was used to send the information to the Data Table. Previously, the output vector ID was the same for each column, even though the output vector titles were correct.



## API

- Corrected issue with feRegenerate, feViewRegenerate, and feViewRedraw where recently created entities did not appear unless a second feRegenerate was called.
- Corrected issue with Message and API font sizes in the preferences. The API was treating them as 4-byte integers but they needed to be 8 byte float.
- Corrected an issue with the AddVisible and RemoveNotVisible methods of the Set object that occurred if these methods were used with the Connection Property, Material, Property or Layup entity types.
- Corrected issue when copying optimization analysis and/or analysis cases containing optimization entities which caused separate references to optimization objects (variables, manufacturing constraints, limits) to not be created correctly.
- Corrected issue that would cause FEMAP to exit unexpectedly when using the GetMeshInfo method on the Beam Calculator object.